• • • **User's Manual**

*el*Master design

a product from

SIGMA
Power Software

elMaster design is one of the main products in the elMaster Power Concept from Sigma Power Software AB. elMaster Power Concept is a comprehensive series of applications with the purpose of making designers' and draughtsmen's work easier and more efficient.

The products included in the elMaster Power Concept are elMaster design, elMaster project, elMaster connect and elMaster interconnect.

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User’s Manual

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Introduction

CHAPTER 1

Congratulations for choosing elMaster design from Sigma Power Software AB. You now have access to a powerful and complete system, which we hope will provide excellent support in your design work.

elMaster design is one of the products in elMaster Power Concept.

In order to be able to use the system fully, you need to know what the system is capable of doing and how to handle it in the best possible way. A good start is to read this manual. Take all the time necessary.

*el*Master Power Concept

*el*Master Power Concept is a series of products with a comprehensive family of applications and programs aimed to rationalize and render the design work for the electrical designer. The products included in the *el*Master Power Concept are:

- *el*Master design
- *el*Master connect
- *el*Master project
- *el*Master interconnect

The products are developed to integrate in networks and have common base functions and working methodology, which will make it easy for you to recognize the different products.

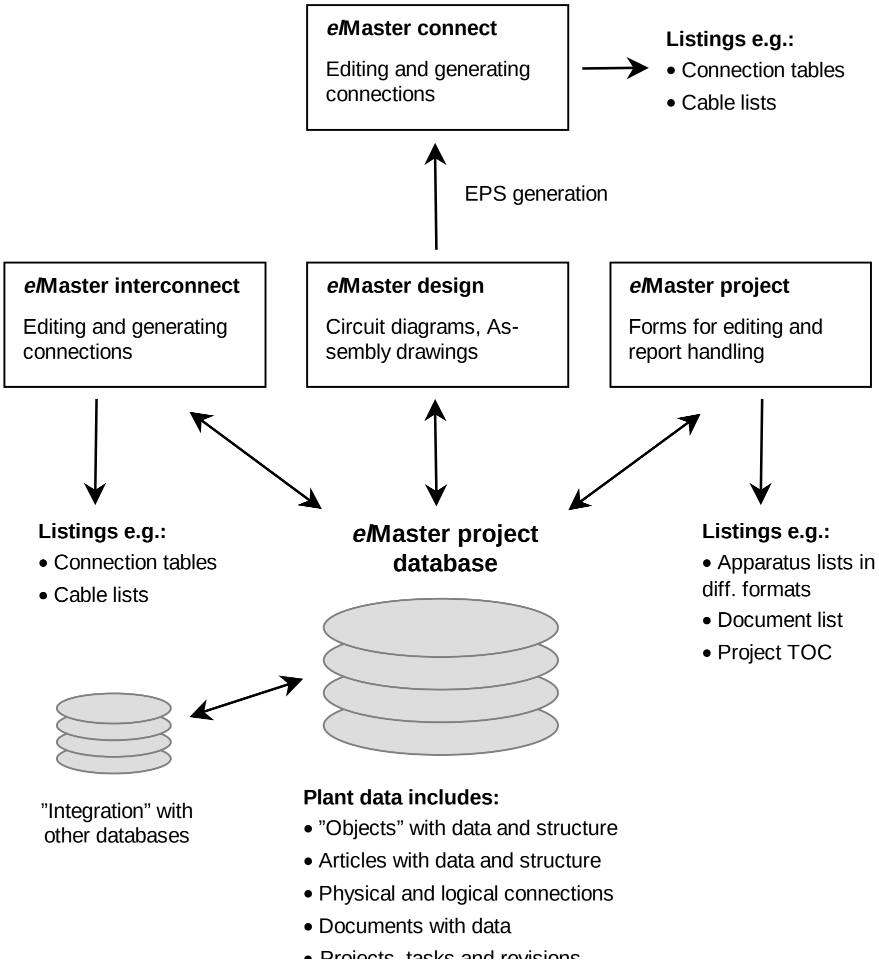

*el*Master connect
Editing and generating connections
Listings e.g.:
• Connection tables
• Cable lists
EPS generation
*el*Master interconnect
Editing and generating connections
*el*Master design
Circuit diagrams, Assembly drawings
*el*Master project
Forms for editing and report handling
Listings e.g.:
• Connection tables
• Cable lists
*el*Master project database
Listings e.g.:
• Apparatus lists in diff. formats
• Document list
• Project TOC
”Integration” with other databases
Plant data includes:
• ”Objects” with data and structure
• Articles with data and structure
• Physical and logical connections
• Documents with data
• Projects, tasks and revisions

Figure 1. elMaster Power Concept

*el*Master design

*el*Master design is based on AutoCAD™ and designed to modernize and rationalize the design process. With *el*Master design lead times can be shortened considerably when more or less complete design solutions easily can be reused. Company standards in the form of ready-made circuit diagrams lead to fewer errors and major gains in efficiency, especially in large organizations with local design offices.

*el*Master design complies the Swedish and European Standard (IEC) where applicable.

*el*Master project

*el*Master project is used in electrical engineering for managing plant structures from simple junction boxes to partial or complete power plants.

Together with other programs in the *el*Master Power Concept, *el*Master project is used to rationalize the design process. All information is stored in a common plant database to facilitate project engineering, operation and maintenance.

The open database structure permits communication with other systems such as maintenance, purchasing and TDM systems (technical documentation management) and article files. *el*Master project saves both time and money for plant owners, consultants and suppliers.

*el*Master connect

*el*Master connect is used for automatic generation of connection tables from circuit diagrams created in *el*Master design.

The connections are presented in various tables and lists adapted to production, installation, and maintenance.

Connections are generated and optimized according to most industry standards to enable simple, efficient installation. The user can also control optimization with his own rules.

*el*Master interconnect

*el*Master interconnect is the new interactive electrical connection program in *el*Master Power Concept.

*el*Master interconnect works with the same database as *el*Master project and *el*Master design. When an apparatus is saved from *el*Master design, the connections between the terminals are also saved in the database.

With *el*Master interconnect you can:

- Change the sew order for a subnet interactively.
- Assign wire numbers manually and/or use automatic rules for assigning wire numbers.
- Assign wire articles manually and/or automatically.
- Assign cable numbers manually.
- Assign cable articles manually.
- Create different cable reports and connection reports.

*el*Master design – an overview

The *el*Master design program contains symbols and advanced commands for creating circuit diagrams, single line diagrams, connection diagrams, and general drawings. Circuits can be drawn in accordance with international standards using portrait or landscape phase lines and in bound or unbound presentation.

Creation of Circuit diagrams

*el*Master design is used primarily for simple creation and revision of circuit diagrams. The objects in a diagram are displayed with handles allowing them to be altered directly on the screen. The program automatically organizes breakup and mending of connecting lines when symbols are inserted or deleted and when entire portions of the diagram are copied, moved or deleted.

During the drawing procedure the symbols can be connected to already existing objects in the database through reference lists. Alternative new database objects can be created directly within the drawing environment.

The user can easily mix connection indication symbols and “draft” symbols in the same diagram. An example of the latter is a symbol where the content of a finished function is to be produced illustratively but without being included in production documents.

Network

*el*Master design has advanced network support for a symbol library and forms as well as for simultaneous generation of connection tables and other lists.

Standards

*el*Master design follows several different standards from circuit diagrams all the way to design databases and generated lists. *el*Master design is a good way of starting to follow standards if these are not already applied. Document life is extended and it becomes easier to exchange documentation with other companies.

Symbol handling is adapted for storing finished circuits and complete diagrams with a clear description, making it easy to spread standards and apply a systematic approach within a company. Advanced network support enables symbol libraries to be stored and updated centrally in a server accessible to everyone. Local libraries can be used for the designer's own symbols. Symbol libraries can be exported and imported for fast distribution between users in major projects.

Automatic Generation

*el*Master design is prepared for automatic creation of production documents from circuit diagrams such as connection diagrams and equipment lists.

With open and documented interfaces, *el*Master design can be connected to the user's own programs or to products within the *el*Master Concept's product family. No extra work is needed to prepare for generating connection documentation, an advantage of the integrated EPS module, which automatically checks the circuit diagram and indicates possible errors.

*el*Master design - for Every Phase of a Project

*el*Master design can be used early in a project to create customized symbol libraries, make sketches and insert finished design solutions. General changes, such as altered designations, form changes etc., can be made automatically on a complete set of documents in order to be able to reuse finished designs.

At the end of the design phase, *el*Master design is used to create connections, mark objects with item designations and analyze each document for errors prior to table generation.

User interface

This section explains the different parts of the user interface and a brief look at how they function and how they are used.

Toolbar

A *Toolbar* is a moveable field that contains various commands. In Figure 2 you can see *el*Master design's own *Toolbar*, which include the alternatives that are most often used when drawing a circuit diagram.

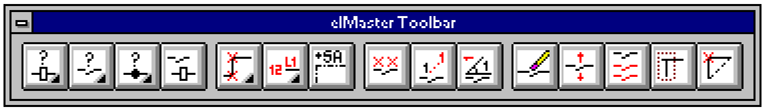


Figure 2. elMaster Toollbar

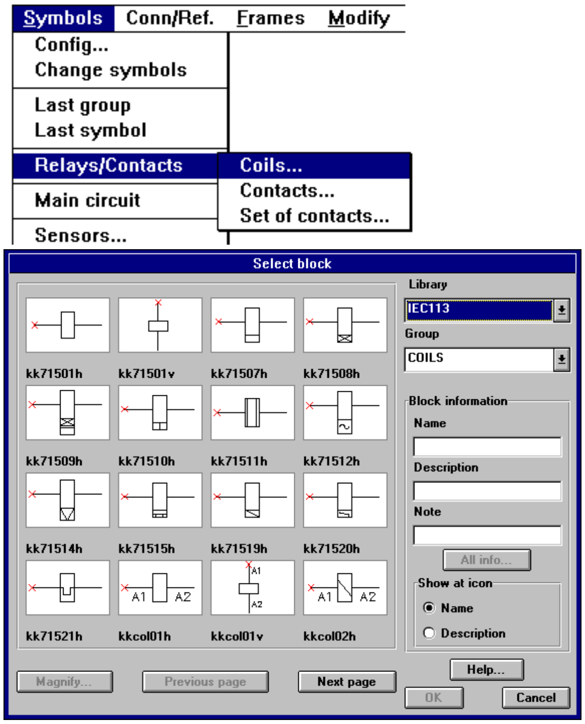
Pull-down menu

A pull-down menu is a menu that can be "pulled down" from the top edge of the screen. With certain graphic systems the cursor must be placed at the top edge of the screen for the pull down menu bar to appear. Pointing at an item in the menu bar shows the pull-down menus.

Note: On certain graphic systems, the titles of the pull-down menus are not visible until you move the cursor above the edge of the drawing.

Icon menu

Icon menus are mostly used when you generate symbols. This type of menu is used because it is easier to describe symbols graphically than it is to describe them with words. The icon menu is called up from another type of menu, usually the pull-down menu. If you select **Coils** from the pull-down menu, as shown in Figure 3, you will receive an icon menu which shows the different types of coils you can choose from.



Library: IEC113

Group: COILS

Figure 3. Icon menu for choosing a coil.

Dialog box

Is similar to the Icon menu, which was described earlier. However, the dialog boxes do not usually include pictures (or icons). The advantage with a dialog box is that it can present the alternatives, which are available to the user. There are many different purposes with a dialog box, and their appearance and build depend on their purpose.

Screen menu

Is the menu that is found on the right hand side of the screen. This is called the screen menu because it is the original screen menu alternative found in AutoCAD.

The screen menu also includes numerous alternatives, which should be seen as complimentary to the other menus. This menu is a hierarchy built menu, which means that you sometimes must “click” several times before you see the alternative you require. If you find yourself “deep” in the menu, you can quickly return to the “top” by pointing at the text **elMaster**.

Cursor menu

Is similar to a pull-down menu, which you activate with a combination of keyboard and mouse or cursor button operations.

This is what you do to display the cursor menu:

1. Hold down the SHIFT-key and press the right hand mouse button, or alternatively, the cursor’s ENTER-button.
2. Hold down the CTRL-key and press the right hand mouse button, or alternatively, the cursor’s ENTER button.
3. Hold down both the CTRL- and the SHIFT-key and press the right hand mouse button, or alternatively, the cursor’s ENTER-button.

Digitizer menu

Menus placed onto a menu plate (the digitizer) with numerous squares that display pictures (icons) or text. Each square corresponds to a function.

Drafting area

Is the area of the digitizer menu that contains no commands. This area represents the screen. The area to the right side of the broken line represents the screen menu.

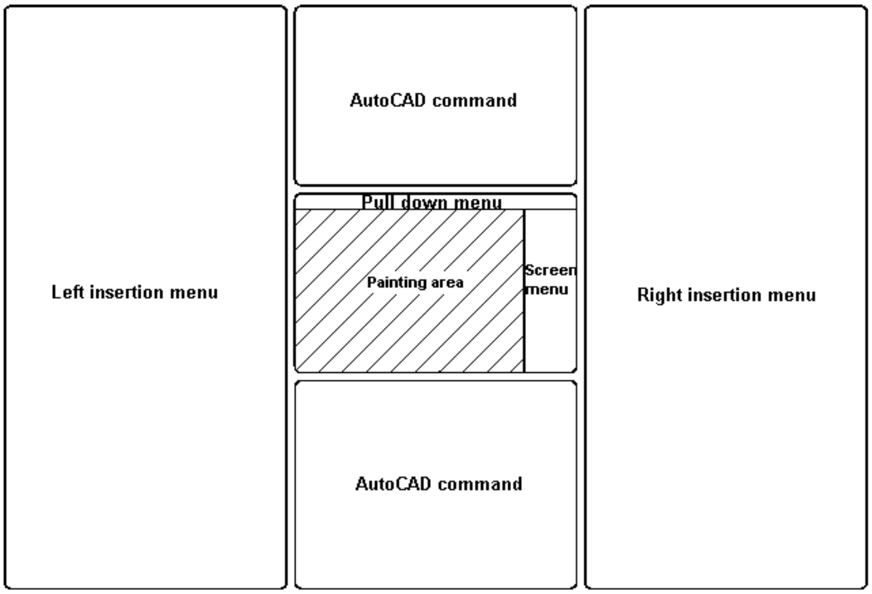


Figure 4.Drafting area

Conventions in the manual

In this manual, the following typographical conventions are used.

Examples of conventions	Description
install	Words or characters that you should type are in bold print. Also buttons that are shown in dialogs, which you should click, are shown in bold print.
apparatus	Italic print is used for labels in dialogs and names of dialogs and windows. Italic print is also used for guidelines.
ELMPROJ.EXE	Words printed in capital letters are file names or directories.
ENTER	Small caps are used for key names or key combinations e.g. ENTER, DELETE and CTRL+V
CTRL+V	A plus (+) between key names states that this is a key combination. This means that you should keep the first key pressed while you press the second one.
File; New project	A choice from a menu is given in bold print. The text on the right side of the sign **;** tells you that it is a sub menu.

Installation Guide

CHAPTER 2

This guide describes the installation procedures for elMaster design. A description of how you update earlier versions of elMaster design is also included.

The following guidelines assume that AutoCAD has been correctly installed.

Installation of *el*Master design

Below are some different methods of installing *el*Master design described, depending on how you will use the program.

Single Workstation Installation

Work procedure:

1. Install AutoCAD in accordance with the guidelines for AutoCAD.
2. Insert the *el*Master installation CD-disk.
3. Start the installation program by double clicking on the file SETUP.EXE in the file manager. Use the alternative **Full installation** and follow the instructions in the installation program.

The minimum installation is:

Program files, Database files, and at least one of the alternative MLUSER, IEC113 symbols or IEC1082 symbols. Any alternative left out, can later be added to the installation.

The installation program will copy all files to the hard disk, add start-up icons to the Program manager and make adjustments in some different configuration files.

Network Installation (Multiple Users)

Work procedure:

1. Install AutoCAD in accordance with the guidelines for AutoCAD.
2. Insert the *el*Master installation CD-disk.
3. Start the installation program by double clicking on the file SETUP.EXE in the file manager. Use the alternative **Full installation** and follow the instructions in the installation program.
4. Adjust the directories for the parts of the program that you want to share in a network, so that these will be located on the server.

All users should have full read and write permissions in both the program code tree and MLUSER-trees.

Workstation

Work procedure:

1. Insert the *el*Master installation CD-disk.
2. Start the installation program by double clicking on the file SETUP.EXE in the file manager. Use the alternative **Workstation** to “connect” a workstation to an existing network installation.
3. Adjust the directories, so that they correspond with the existing directories.

This alternative will NOT copy any *el*Master files to the hard-drive it will only set-up the system files that are needed locally. The alternative will add start-up icons to the Program manager and make adjustments in some different configuration files.

Extra MLUSER tree

After the installation, you can create extra MLUSER-trees, which you can use to store symbol directories and settings. To be able to use a new tree, the search path to the MLUSER-tree should be added to any possible earlier search paths in the MLUSER-tree. The search paths are set in the file MASTLINE.INI.

Preparation for updating a Workstation

As a matter of security, take a complete copy of the existing installation prior to updating to this version.

If you have personal settings in *el*Master, SAVE THESE before you install the new version of *el*Master design. The easiest way to do this is to install the new version in a separate directory and then copy all personal settings and blocks to the new version's directory when the program has been installed. When the new version is operating correctly the old version can be removed.

Note! Install the program code for *el*Master design in a separate directory from that used by earlier versions of *el*Master.

Procedure:

1. Make a backup of the old installation
2. Install the new version in a SEPARATE LIBRARY.
3. Read symbols and symbol-libraries into SYMLIB. See ”Symbols and Converting symbols” on page 32.
4. Read your own forms into FORMAT. See “Converting formats“ on page 33.
5. Move any own created adjustments to the new code.

Symbols

Procedure:

1. Create a new symbol group with the function **New** in the dialog box *Admin Blocklib.* The function will create a directory on the hard disk.
2. Enter description and default-values for the insert-properties for this group.
3. Copy the blocks to the directory with the operating system command.
4. Use the command **Read** in the dialog box *Admin Blocklib* to import the blocks that have been copied.
5. Enter a description and change when necessary the insert properties for each block. Don’t forget to **Save data** for each block.

It is now also possible to create slides to the blocks with the function **Create Slides** in the dialog box *Admin Blocklib*.

Converting symbols

Symbols created in version 12 or in an earlier version of *el*Master design must be converted. This can be done manually with the functions in **Blocbuild**, or automatically for a whole group of symbols.

Depending on how the symbols are made, it may be necessary to do some manual adjustments, even if the automatic converting function is used.

Procedure:

1. Create a report with the symbols that should be converted. The report must be of the type **Standard** and the symbols should not be exploded. The function to create a report of this type is in the dialog box *Admin Blocklib.* Don't forget to unmark the setting **Explode when insert**.
2. Select **Initiate blockbuild**.
3. Start the function **Convert Blocklib**.
4. Explode all symbols. Make sure that all parts of the symbol is inside its frame. If not, move all parts of the symbol inside its frame.

5. When all adjustments in the report are made, the symbols should be imported to the disk. This can be done with the function **Import** in the dialog box *Admin Blocklib*.

Converting formats

Formats (*.FRM files) created in version 12 or in an earlier version of *el*Master design must be saved as blocks.

Procedure:

1. Use the function **As block** under **Format** in *el*Master 12. The block is saved in the same library and with the same name (but with the file extension DWG) as the form definition.
2. When the new version has been installed, use the function **Admin** under **Format** in the screen menu. Create a new symbol group with the function **New** in the dialog box *Admin Format*. This will create a directory on your hard disc.
3. Enter description and the default-values for the insert-properties for this group.
4. Copy the forms to the directory with the operating system command.
5. Use the command **Read** in the dialog box *Admin Format* to import the blocks that have been copied.
6. Enter a description and change when necessary the insert properties for each block. Don’t forget to **Save data** for each block.
7. It is now also possible to create slides to the blocks with the function **Create Slides** in the dialog box *Admin Format*.

User adjustments

If you have created your own functions or made any additions or adjustments in any of *el*Master's user adjustable files, use a text editor to copy these adjustments to the corresponding file in the new installation. Be sure to document any changes.

Menu file

If you made any changes to the menu file, these should be copied to the new menu file. Note that the syntax control that takes place when AutoCAD compiles the menu, is stricter than in earlier releases. This will mean that there may be rows in your menu file that no longer will work.

Functions

These are the files in *el*Master that may be changed:

File	Description
*.MNL	Start-up file
*INIT.LSP	Initiate functions and variables
*CMD.LSP	Adjustments and re-definitions of commands
*MNUDEF.LSP	Functions used to define the user interface

These files are placed in the MSC-directory. With a normal text editor, for example “notepad”, changes in these files can be copied to corresponding file in the new installation

Configure

User configurations can either be an adjusted prototype drawing or adjustments in the CFG-files in the CFG-directory.

Changes in the CFG-files must be copied to the new file.

Configure AutoCAD and *el*Master design

The actions, described below, are normally done by the installation program. However, you might have to do this manually if parts of the program have been moved or copied.

Note that all directories that are given below are examples and have to be changed to valid directories.

Configure AutoCAD for your hardware

AutoCAD must be configured for the hardware that it is used with, for example printers and digitizers. This information is saved in a CFG-file. For further information, see the documentation for AutoCAD.

Which configuration file that should be used can be controlled by the command line switch “/c”.

Configure AutoCAD for *el*Master design

To get AutoCAD and *el*Master design to work properly together, a number of settings must be done. These settings are save in a so-called profile. Each profile has a name and is liked to each separate installations of AutoCAD. For further information about profiles, see the documentation for AutoCAD.

The command line switch “p” can control which profile that should be used. The installation program will automatically add a profile with the name **elMaster**. On the CD for *el*Master, there is a program, REGISTER.EXE, which will create a profile in the same way as the installation program. The profile must contain the following settings:

- **Support File Search Path** must contain the path to the MSC-directory
- **Template Drawing File Location** must be set to the MSC- directory.
- **Menu File** must be set to the file SOSED in the MSC- directory.
- **Alternate Font File** must be set to an existing font file e.g. SIMPLEX.
- **Reload AutoLISP between drawings** must be marked.

System files

A number of DLL-files, which only can be added by running the installation program, are needed to get *el*Master design to work,

Symbol libraries and MASTLINE.INI

These settings are saved in the file MASTLINE.INI, which is located in the same directory as the file ACAD14.CFG or ACADNT.CFG (depends on which version of AutoCAD you use).

Work procedure:

1. Select **MLCONFIG; SET ini** in the screen menu.
2. Add path to the program-tree and the MLUSER-trees, e.g.: Program code: c:\elmaster\diagram MLUSER1:C:\ELMASTER\MLUSER;C:\ELMASTER\IEC113;C:\ELMASTER\ IEC1082
3. End the dialog box with **OK**.

Remark: It is possible for you to use more than one MLUSER-tree at the same time by specifying more paths, separated with ”;”. In the dialog box, *SET ini*, there is room for up to tree different combinations. The first (MLUSER 1) is always default.

ODBC

The functions to handle symbols and drawing frames are using information that is stored in a database. To read this information, a connection of the database must be done from AutoCAD. This connection is done with ODBC. The DATA SOURCE NAME for the database must be SOS_BLOCKLIB, its driver must be of the type ”Microsoft Access”, and the database file BLOCKLIB.MDB.

• • • Getting Started

CHAPTER 3

This section is not a complete training guide of elMaster design, more a general introduction to the program. We will with a minor drawing example show you how to draw and use some of the different features in elMaster design.

We are going to draw a simple circuit diagram, which eventually will look like Figure 5.

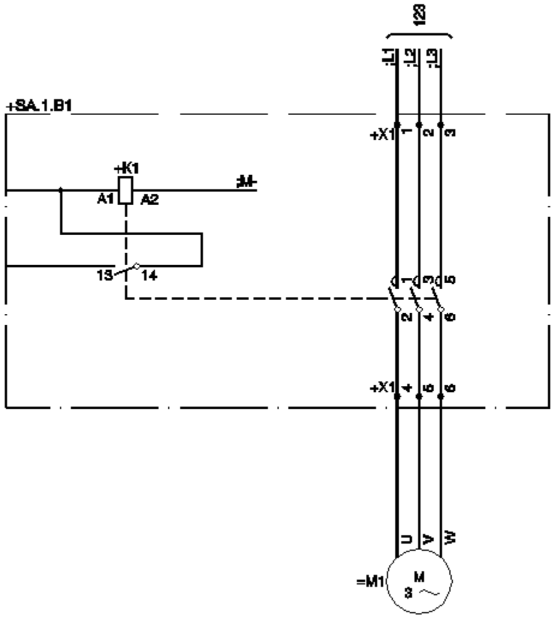


Figure 5.Circuit diagram

The icons, which are shown in some sections, refer to elMaster Toolbar.

Insert a form

We will start by inserting a form (drawing frame).

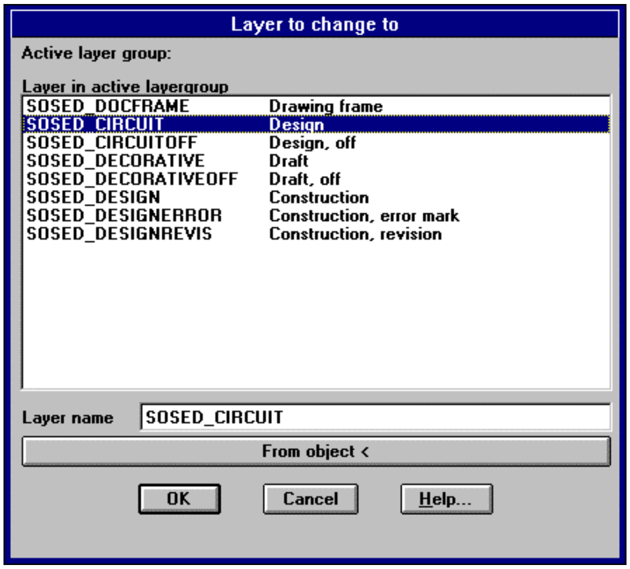
1. Choose **FORMAT** from the screen menu (you might have to select the text **elMaster** to find **FORMAT**) and then select **Quick.**

The *el*Master's standard drawing frame should now be visible.

Check that you have the correct layer available

In *el*Master design you draw in different layers to simply differentiate the various information. You are now going to draw in the layer SOSED_CIRCUIT.

2. To make sure you have the right layer, select **Modify** in the pull-down menu and then **Layer**. The dialog box shown below will appear.



Layer to change to

Active layer group:

Layer in active layergroup

SOSED_DOCFRAME	Drawing frame
SOSED_CIRCUIT	Design
SOSED_CIRCUITOFF	Design, off
SOSED_DECORATIVE	Draft
SOSED_DECORATIVEOFF	Draft, off
SOSED_DESIGN	Construction
SOSED_DESIGNERROR	Construction, error mark
SOSED_DESIGNREVIS	Construction, revision

Layer name SOSED_CIRCUIT

Figure 6.Layer to change to

3. Highlight the layer you want and confirm that the name of the layer is given in the *Layer name* box.
4. Click on **OK**.

Draw the phase lines of the Main Circuit

We are now going to do the layout of the phase lines of the main circuits.

5. Select **Conn/Ref; Main circuit; Multiple connection lines**.

*el*Master will ask you questions regarding the starting point, the start pitch, endpoint, the end pitch and the number of connection lines. A simple explanation of this is given in Figure 7.

6. Point in the same order as shown in Figure 7. Make sure there is a distance of 5 mm (2 snap) between the lines.

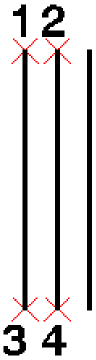


Figure 7. Phase lines of the main circuit

Draw the Boundary Frame

The boundary frame should be of the dash-dot type.

7. Select **Frames** and the alternativ “. - “.
8. Set the upper left point of the frame and then set the frames lower right point.
9. Fill in the location designation **SA.1.B1** in the dialog box and press **OK**.

Create the Main Contacts

10. Select **Symbols**; **Main circuit; Power circuit switch**. A dialog box appears.
11. Mark the symbol **Contactor** by pointing at it. Then click **OK** (alternatively, double click on the symbol).

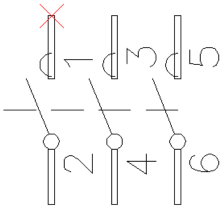


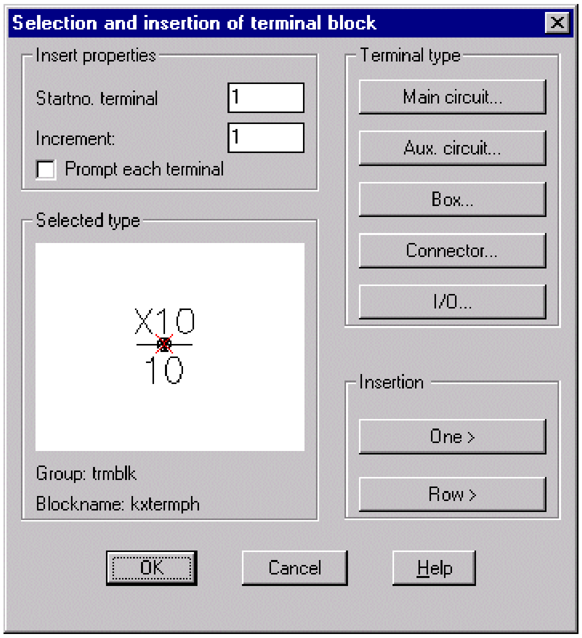
Figure 8. Contactor

12. Place the symbol in the correct position, see Figure 5.
13. In the following dialog box, click **OK**.

Note that the phase line of the main contact breaks up automatically.

Create the Terminal Rows

14. Select **Symbols; Terminal blocks; Type**. The dialog box shown in Figure 9 should then appear.



Startno. terminal: 1

Increment: 1

☐ Prompt each terminal

Group: trmblk

Blockname: kxtermph

Figure 9. Selection and insertion of terminal block

15. Select **Main circuit**. A dialog box will appear where you can select different symbols that are available for main circuits.

It is important that you select the correct type of main circuit, especially if you intend to create connecting tables. As you can see, there is a box line for every type of terminal. This specifies where in the frame the terminal will be positioned. You can also select different forms of terminals (with points, circles or just a line). We shall start with the upper terminal row (X1:1-3).

16. Select the terminal **External up** as shown in Figure 10.

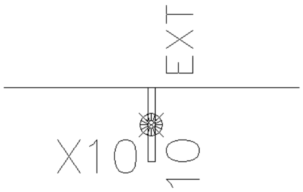
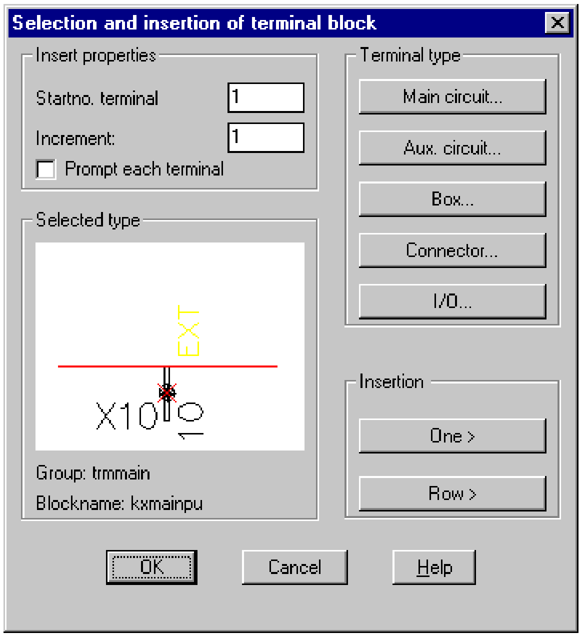


Figure 10. External up

We shall now return to the first dialog box shown in Figure 9. Here we shall give the parameters of the designation of the outlet of the terminal row. In this example, the outlet of the terminal row shall be 1, 2, and 3 in numerical order from the start number and upward and in increments of one. As the terminal rows are in numerical order, we do not need to select **Prompt each terminal**. This means we cannot jump (numerically) when we are creating a terminal row. If the outlet had been, for example 1,5 and 11, then we would of have to select **Prompt each terminal**.

17. Set the parameters as shown in Figure 11 and then select **Row**>.


Selection and insertion of terminal block

Insert properties

Startno. terminal: 1

Increment: 1

☐ Prompt each terminal

Selected type

Group: trmmain

Blockname: kxmainpu

Figure 11. Selection and insertion of terminal block

We are now going to insert a row of terminals on the main circuit lines. To do this you only have to specify two points. These points should make a line that crosses the circuit lines where the terminals should be placed. Note that the insertion point for the terminals is a "snap" inside the frame.

18. Set "First point" to the left of the main circuit lines and a "snap" inside the frame. Set "Second point" to the right of the main circuit lines and a "snap" inside the frame.
19. Press ENTER, to end the function.
20. In the dialog, which will appear, common data for the terminals is written. Insert the positioning designation X1.

We are now going to insert the lower terminal row (see Figure 5), the row X1:4-6. Before you insert the terminals you have to choose another terminal type.

21. Select **Main circuit** and then the terminal type **Extern down**.

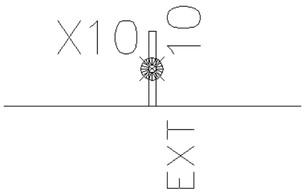
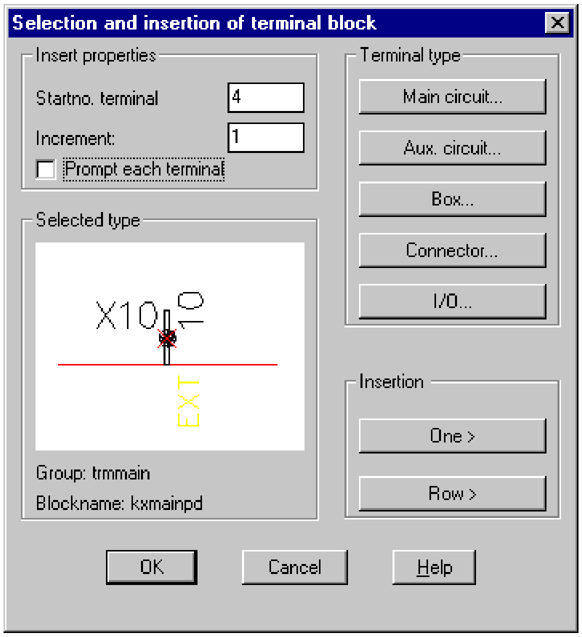


Figure 12. External down

The dialog box should look like Figure 13 before you continue with the next terminal row.



Startno. terminal: 4

Increment: 1

☐ Prompt each terminal

Group: trmmain

Blockname: kxmainpd

Figure 13. Selection and insertion of terminal block

Create the References

We shall now look at sheet referencing at the top of the circuit (in this case, L1, L2 and L3).

22. Select **Conn/Ref; Ref.Row; General reference**.
23. *el*Master wants you now to put the end points in a frame. Window mark the connection, which the reference will be located on.
24. The next task is to decide which sheet you are going to refer to. Give 121.
25. If you look at the drawing, you can see a cross on the left-hand wire and *el*Master prompt **Signal/Reference**. Give L1. The cross will now move to the next wire. Give L2, and do the same again for L3.

*el*Master will now create the reference, complete with the bracket/clamp.

Create the Motor

26. Select Symbols; Main circuit; Motor.
27. Select the motor as shown in Figure 14.

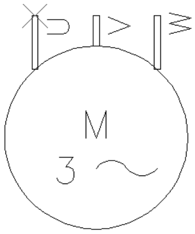


Figure 14. Motor

28. Position the symbol at the bottom of the main circuit.
29. Give the designation M1 under the function designation.

Create the Contactor Coil

30. Select **Symbols; Relay/Contact; Coil**.
31. Select the coil as shown in Figure 15 and press **OK**.

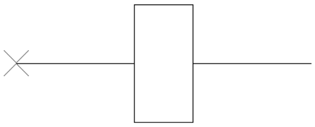


Figure 15. Coil

32. Position the coil correctly in the drawing, see Figure 5.

33. Fill in the function designation K1 and the outlet designation A1, respective A2.

Create the Contact

34. Select **Symbols; Relay/Contact; Contact**.
35. Mark the contact as shown in Figure 16 and press **OK**.

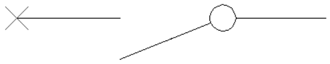


Figure 16. Contact

36. Position the coil correctly in the drawing, see Figure 5.
37. Fill in the outlet designation 13 and 14.

Create the Mechanical Link

38. Select **Conn/Ref; Mech.Link**.

When you have selected this command you will automatically get a “pickbox” (AutoCAD’s objectsnap “Node”). This means that *el*Master is searching for a hidden point, which is in-laid in several symbols. These are used to latch the mechanical link. The reason for this is to find a point, which is not found in the command SNAP.

39. In the coil, there is a point in-laid. Start the link here. Make sure that this point is available in the pickbox when you select this command. Draw this link, as it would a normal line, through the contact and then angle the link towards the main contact.

Make the Connections

40. Select **One connection line**
41. Pull the connection from the edge of the box directly over the coil as show in Figure 17. Note that the connection breaks up when you draw over the existing symbol.

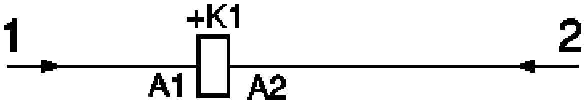


Figure 17

42. Repeat this command by pressing ENTER (or the equivalent command with the mouse-button) and draw as shown in Figure 18. Ortho should in this case be off. As you can see, you now get partly a connection point and partly an angle deviation.

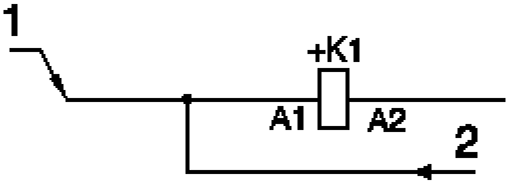


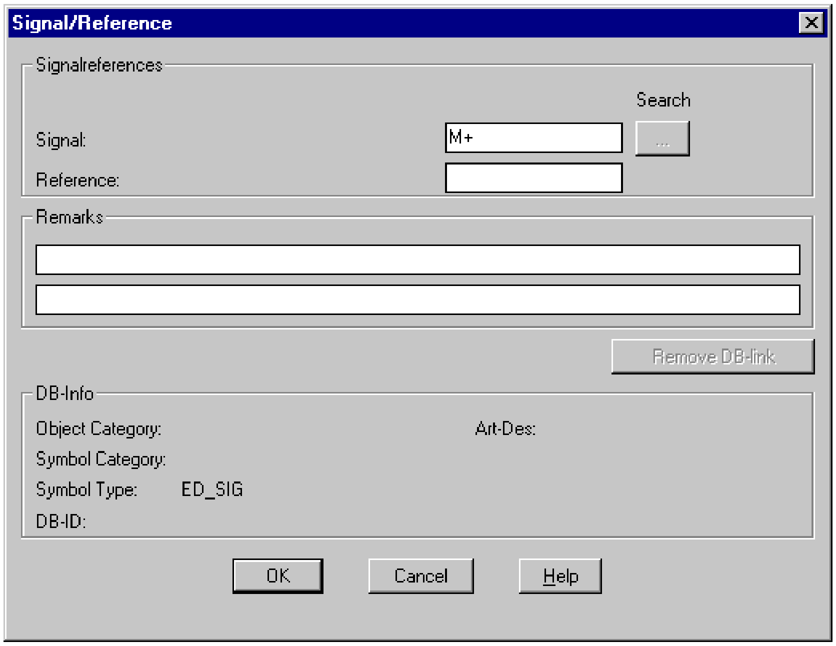
Figure 18

43. Repeat this procedure and pull the vertical part of the junction directly over the contact and out to the edge of the box.

As an alternative, you can try the command **Free connection**. The difference between these two functions is that with **Free connection** you have to pick each corner in the connection, whilst you **with One connection** only pick the start- and endpoint.

Create the Reference M+

44. Select **Con/Ref; One ref.; General Ref**.
45. Point the cursor at the end point of the line and fill in “M+” in the dialog box as shown in Figure 19.


Signal: M+

Symbol Type: ED_SIG

Figure 19. Signal/Reference

Your drawing should now like the drawing shown in Figure 5.

*el*Master includes several useful editing functions. We are now going to look on them.

Special Editing Functions

The editing functions can be used for deleting, moving or copying part of a drawing. The advantage with these functions is that they automatically maintain control of a line. If a line becomes located below a symbol after copying or moving, it is removed automatically and then returned after subsequent moves or deletions. If you want to add a “bend” to a connection line, then use the function **Break connection**. This function breaks the line and adds a “Node” to this point. This command is found in the pull-down menu under the title **Change**. Alternatively, you can use **Ed_Erase**, **Ed_Move**, **Ed_Copy** and **Break Connection**.

It is important to remember that the commands **Ed_Erase**, **Ed_Move** and **Ed_Copy** are not the same as the AutoCAD’s commands with the same name. The command **Stretch** is AutoCAD’s command.

Ed_Erase

46. Select **Ed_Erase** and point the cursor at the contact. As you can see, the line now becomes joined.

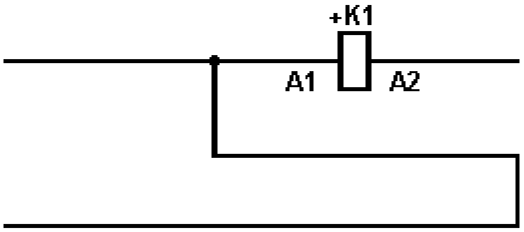


Figure 20

Ed_Move

47. Select **Ed_Move** and point the cursor at the coil. Set the base point and move the coil to the left. Note that even the break-up point is moved.

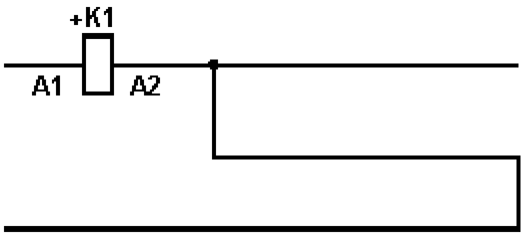


Figure 21

Ed_Copy

48. Select **Ed_Copy** and point the cursor at the new coil. Set the base point and copy over to the line below.

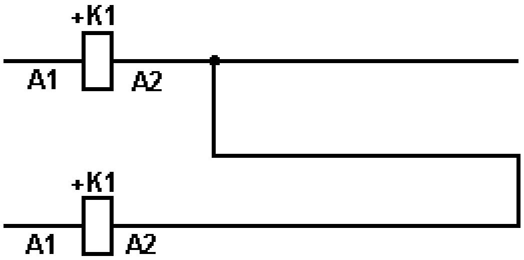


Figure 22

There is also a function called **Ed_Copy + Edit**. This does exactly the same as **Ed_Copy** but also displays, in the dialog box, the values of each symbol that have been copied.

Break Connection

49. Select **Break Connection** and point the cursor at the lower connection line. Pull the line downwards.

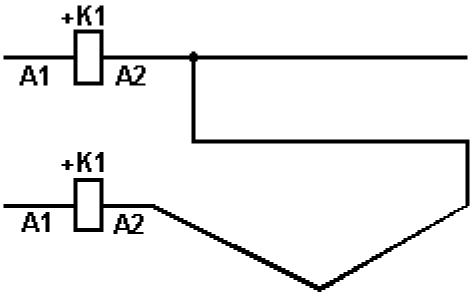


Figure 23

50. To make sure the angles in the diagram are correct, repeat the command and point the cursor the line on the left-hand side of the previous point. Pull the line up and to the right.

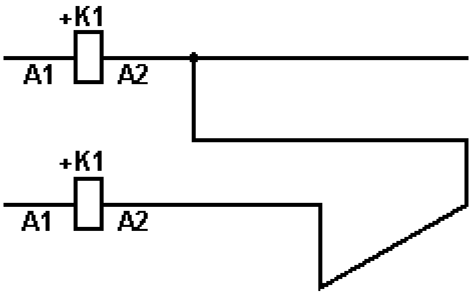


Figure 24

Finally, the last angled line should be corrected.

51. Repeat the command and point the cursor to the right hand corner. Then pull the line down.

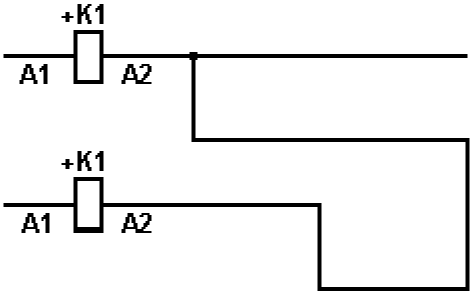


Figure 25

CHAPTER 4

elMaster design is divided into three parts:

- ***Diagram***

  Is the base in elMaster design and is used for drawing circuit diagrams and other documents not according to scale, for example single-line drawings.

- ***Layout***

  Is the function that supports production of drawings according to scale, especially assembly drawings, which occasionally are referred with cubicle layouts.

- ***EPS***

  Is the check function, which check circuit diagram sheets prior to the ensuing generation of connection tables.

This chapter describes in general how elMaster Diagram and Layout works. Among other things you can here read about how you use forms, layers and symbols.

A description of elMaster EPS you will find in a separate chapter.

For more information about the various commands, refer to the Command Reference.

Starting *el*Master design

*el*Master design is an additional application of the design program AutoCAD and is specially developed for electrical design. The program consists of a number of functions and symbols (blocks). You will automatically get access to these if you have installed AutoCAD and *el*Master design correct, see the Installation Guide.

When you start *el*Master design *el*Master's menu file is loaded into AutoCAD. Information about which menu file that should be loaded is retrieved from the configuration file in AutoCAD. The menu file refers to a so-called MNL-file, where instructions for the loading of *el*Master's functions can be found. At the initialization a control is made to see which functions that should be loaded, *el*Master Diagram or *el*Master Layout. If this information can not be found (e.g. when a new drawing is created), you will be prompted about it.

Forms

Forms are used for delimiting a drawing area and in a clear way show information about the drawing, for example who has drawn it, firm and title of the drawing etc. This data is defined in a dialog box, which appears after inserting a form. The information will be displayed in the revision head, which is normally placed at the bottom left corner of the drawing frame.

*el*Master design has several predefined forms but you also have the possibility of making your own form definitions. The forms are divided into groups (libraries). Each definition corresponds to a library in an MLUSER-tree.

Reference net

To be able to refer to a specific part in the drawing the forms are equipped with a reference net. The reference net is shown as a coordinate system in the marginal of the drawing.

With the help of the dialog box *Administrating Formats* you can configure the reference net. There are four fields; Ref, size, Pos, Char.

The field Ref contains information about the number of squares in the grid on the format X, Y, for example Rithuv3 have the value 6,4 meaning that this drawing frame has six squares in X-direction and four squares in Y-direction.

The field Size contains information about the size of each square in the format X, Y, for example Rithuv3 have the value 70,67 meaning that the size of the squares in this drawing frame is 70 mm in X-direction and 67 mm in Y-direction.

The field Pos contains information about the position (offset), in the format X, Y, between the reference net's origin of coordinates and the insertion point (coordinate 0,0).

The field Char contains information about the direction of the characters. A positive character states that the lowest number or character is in the upper left corner of the drawing, see Figure 26 where the field Char is set to A,1. A negative character states that the highest number or character is in the upper left corner of the drawing, see Figure 27 where the field is set to –A,-1. In Figure 28 the field is set to A,-1.

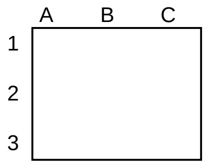


Figure 26

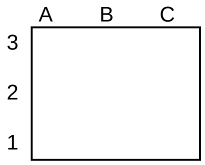


Figure 27

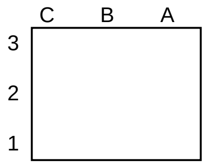


Figure 28

Network support

The functions handling the drawing forms are written to operate in a network environment in which several users share the same settings. The libraries are locked when they are being used so that a user should not be able to modify the contents while another user is working with the library.

Insert form

With the command **Insert (FORMAT)** you insert a form. In the dialog box, which appears with the command, you choose which form definition (library) the form should be selected from.

You also choose which format type the form should have. Examples on common form types are A3 horizontal and A4 vertical. The format type is registered by the PLOT-function.

If the form should be inserted as a block you can choose whether the block should be exploded during insertion or not.

For more information about the command see page 207.

Create form definition

With the help of AutoCAD's lines and *el*Master's attribute definitions you can draw you own forms. You save the forms with the command **Admin(FORMAT)**. With this command you can create own form definitions.

For more information about the command see page 138.

Layers

When you draw in *el*Master design you can use different layers to distinguish certain information. For example there are layers where you draw "design" i.e. the layers are interpreted by the system as "design". Other layers you use as "draft". Draft layers are the opposite of "design", which means that connections not will be included in connection tables.

Entities in the program always belong to a layer. Normally these entities have the same properties as the layer to which they belong to, for example type of line and color. If you move an entity to another layer, the entity changes its line type and color to match that of the new layer.

You can specify the way entities should behave by giving them the line type BYLAYER, which changes according to its environment. If you want an entity to belong to a specific layer but have its own unique line type, you can specify the line-type for the entity with help of the commands under the **Change**-menu. The entity can then have a constant line type regardless of which layer it belongs to. The same applies to colors.

In the pull-down menu **View**, there are several commands for managing the layers in *el*Master design.

Important
It is important for all users of this program (within the same company or group) to decide which colors are to be used in the different layers. This is to prevent misunderstandings when drawings are sent to different departments or work places. Because drawing documentation has a long life span, we recommend that you do not change the basic configuration.

Layer system

*el*Master design works with a layer system that contain base groups with layers, which can further on be divided into layer groups. *el*Master Diagram only use a base group that consist of those layers that make up the base in the program. *el*Master Layout consist of both base groups and a number of layer groups.

You have the possibility of creating your own base groups and layer groups.

The layer system is described in a file named LAYERS.CFG. This file is registered when you create a new drawing and creates a layer system according to the specification in it.

The layer system is flexible and can be adjusted according to your particular needs. However, we recommend that you follow the layer structure delivered as standard for respective applications.

The base group

A base group consists of a number of layers and is used as a base in all layer groups. Further down in this section you can see which base groups and layer groups *el*Master design (Diagram and Layout) consist of.

For each layer in the base group the description is saved in the form of a text line. This description is printed out together with the layer name when you select a layer.

If you create own layers use relatively short names in the base group.

The layer group

Layer groups are only used by *el*Master Layout and is used to easy turn on/off several layers at the same time. A layer group can for example consist of apparatuses. By selecting the command **Layer status** or the commands under **LGROUP** in the screen menu you can turn on/off the layer group, i.e. all the apparatuses in the drawing will be influenced by the command (turned on/off).

You can create own layer groups by adding a group prefix (any number of characters) before the layer names in the base group. All layer groups have the same number of layers as the base group.

To each layer group a description is saved in the form of a text line. This description is printed out together with the group name when you select layer groups.

Locking of non active groups

If you want to protect objects on other layer groups than the active, you can lock these. When you change the layer group the application automatically locks the layers in the old layer group and unlocks all the layers in the new active layer group. In the configuration you set if you want automatic locking or not.

Configuration of the layer system

The file LAYERS.CFG contains a description of how the layer system is constructed. In the file there is a description of the layers in the base group (name, line type and color), and a description of the layer groups. LAYERS.CFG also contains information about the active layer group.

New drawing

When you start a new drawing or edit an existing drawing that was created outside of *el*Master design, the application creates a new layer system. The application registers the configuration of layer systems from the file LAYERS.CFG in the drawing's directory, or if this file does not exist, for the file LAYERS.CFG in the directory CFG in the program code tree. The layer system is created according to the settings in LAYERS.CFG. The layers in the layer system that are already defined in the drawing do not change properties (color, line type).

Layers in *el*Master Diagram

The layer system in *el*Master Diagram consists only of a base group. The table below shows the layers that are standard in *el*Master Diagram.

Layer name	Description	Color
0 (Zero)	AutoCAD's layer 0 (must be found)	7 (white)
SOSED_CIRCUIT	"Design" layer	7 (white)
SOSED_CIRCUITOFF	"Design" layer, hidden information	1 (red)
SOSED_DECORATIVE	"Draft" layer (not "design")	4 (cyan)
SOSED_DECORATIVEOFF	"Draft" layer (not "design"), hidden information	1 (red)
SOSED_DESIGN	Designer's layer	2 (yellow)
SOSED_DESIGNERROR	Error indication layer (with sheet control)	1 (red)
SOSED_DESIGNREVIS	Layer for revision markings	1 (red)
SOSED_DOCFRAME	Layer for drawing frame	7 (white)
SOS_DOCCONFIG	Layer which elMaster uses, not for use by user.	7 (white)

SOSED_CIRCUIT

Used for drawing design layers. For example, connections and symbols placed in this layer are interpreted by the system as "design". The information in the layer is visible.

SOSED_CIRCUITOFF
Is also “design”, the difference being that the layer is normally “turned off”. The layer is used for hidden, “design” information

SOSED_DECORATIVE
Used for “draft” drawing. “Draft” is the opposite of “design”, in other words, the connection symbols will show in the drawing (in another color) but will not appear in the connection table.

SOSED_DECORATIVEOFF
The same as “draft” except that it is normally “turned off”. For example for the designer’s comments.

SOSED_DESIGN
Is the so-called designer’s layer. This is used to incorporate information into a circuit schematic sheet, which does not normally belong to the documentation.

SOSED_DESIGNERROR
Used by *el*Master for error indication with, for example, sheet checking during the table generation process.

SOSED_DESIGNREVIS
Used for the incorporation of revision markings.

SOSED_DOCFRAME
Is the layer in which the graphical objects of the drawing forms should be drawn. Includes the possibility to “turn on or off” the title block of the drawing. The drawing form is normally “turned off” when plotting on pre-printed forms.

SOS_DOCCONFIG
Includes the configurations of the drawing. This layer is used by *el*Master and is normally of no interest to the user.

Layers in *el*Master Layout

The layer system in *el*Master Layout consists of several layer groups. Each layer group consists of a base group with different layers.

The layer groups in *el*Master Layout are following:

Layer group	Prefix
Enclosure/Cubicle	C
Group/Plates	P
Apparatus	A
Hole punch	H
Label/Sign	S
Design aid	D

Each layer group contains the following base group:

Layer	Description
01	Continuous, thick
02	Continuous, fine
03	Dashed, thick
04	Mounting area. Dashed, fine
05	Center line. Dashdot, fine
06	Dashdot, thick
07	Double dashdot, fine
08	Dimension, line
09	Pos. no.
10	Hatch
11	Text, fine
12	Dimension, text
13	Text, thick
14	Design
15	Drawing frame
16	Information

Line types and line widths

*el*Master Diagram has a number of its own line types for drawing circuit schematics. These line types are not available from the start, but are loaded when a boundary frame or mechanical link is drawn. The line type can be manually loaded-in from the file SOSED.LIN.

Remark: In *el*Master Layout, only AutoCAD’s own line types are used.

*el*Master’s line types:

Name	**Description**	**Used for**
SOSED_MLINK	_ _ _ _ _ _ _ _ _ _ _ _ _ _ _	Mechanical links
SOSED_FRAMEDOT	____ . ____ . ____ . ____	Boundary frame dash-dot
SOSED_FRAMEDDOT	__ . . __ . . __ . . __ . . __	Boundary frame dash-double dot
SOSED_FRAMECONT	____________________	Boundary frame continuous

In *el*Master diagram, it is possible to use two different line widths. A normal width is 0.0 mm on the screen, which will then normally give a width of 0.25 mm during the plotting procedure. There is also a thicker width available for main circuits, which is already assigned on the screen.

*el*Master has a pre-set line width of 0.5 mm for main circuits. This can be changed, but is not recommended. This is because the distance between the terminal and the lines will not be correct. According to specified standards, the line width for main circuits must be double to the width of normal/secondary lines.

Symbols

In *el*Master design there are a large amount of symbols.

The symbols fall broadly into four categories:

- **"Design" symbols**
  Information from these can be used to generate connection tables etc.

- **"Draft" symbols.**
  Do not include information for connection generating, but are used to more thoroughly specify a function.

- **Reference symbols**
  Used to indicate where another symbol is located, for example contact trees.

- **Compound symbols**
  Can be complete supplies or parts of a standard circuit, which are a compound of the symbols given in the categories and connections above and are not specified in any standard. These symbols usually "explode" when they are inserted.

The symbols are defined according to the older IEC 113 Standard and to the newer IEC 1082-1 Standard. You can specify which standard you wish to work with in the configuration.

Roughly you can say that the symbols according to IEC 133 have terminal designations located under the connection lines, whilst IEC 1082-1 have terminal designations located above.

Insertion of symbols

To insert a symbol you can use the functions under **Symbols** in the pull-down menu or the function **Insert** under BLOCKLIB in the screen menu.

All the connection-giving symbols supplied with the product can be placed on top of an existing conductor. The part of the conductor covered by the symbol is automatically deleted. For this to occur, the symbol must include a terminal or node (POINT).

A more detailed description about how you insert a symbol you will find in the Command Reference on page 205.

Administration of symbols

You also have the possibility to create and administrate your own symbols. You do this with the function **Admin** under **Blocklib**. From here you also have the possibility of creating reports and importing blocks from a standard report to a group in BLOCKLIB.

Data can be partly connected to a group and partly to individual blocks. Every group has certain insertion properties, which apply to all the blocks in a group. If you specify properties for an individual block, these properties will override the original properties of the group.

Designations for symbols

Item designation, terminal designation and other data for symbols are handled with the help of the *Object* dialog box (is shown after inserting a symbol). This dialog box looks different depending on the symbol type.

Connect article to apparatus

Remark: *el*Master project is needed to get access to the function **Art**.

With the button **Art** in the *Object* dialog box, you start a function, which can assign an article to an apparatus. If the apparatus has been assigned an object category earlier, this value will be used as a default selection criterion. If this is not the case, you have to choose an object category in the pull-down menu.

With the filter, you can limit the number of articles, which are shown in the list. You can also use so-called “wildcards”, i.e. “?” for one sign and “*” for several signs. For example, to make a list of all the words that start with a “d” and ends with an “a”, write **d*a**.

How to find symbols in *el*Master

It may not always be obvious to you where to find a symbol. To find out which symbols *el*Master consist of there is a function, which is easily can print these out. See “How to create a symbol-report”, page 67.

The symbols that are printed out also include your own created symbols as these are stored together with those of the systems, provided that you have stored them with the help of the functions in BLOCKLIB.

The symbol library (BLOCKLIB)

BLOCKLIB is the name for the functions that help you handle the symbols. The basic concept behind the symbol library is that you should be able to work with blocks and icons (slides) in a simple and understandable manner.

BLOCKLIB has two dialog boxes, one for inserting the blocks and one for administering them.

There are functions to create and erase specific blocks. There are also functions for registering and creating icons (slides) for blocks drawn with other AutoCAD applications. You can, for example, register symbol libraries from a supplier of details and then select among these in the same way as when you are working with your own blocks.

Libraries

Every block belongs to a block group. All the blocks in a block group must be in the same directory.

When you create a block group, a corresponding block group library is created as a secondary directory to the directory BLKSLD in the MLUSER-tree. You can have an optional amount of block group libraries in every MLUSER-tree.

The library name must be a valid directory name and must not include more than 32 characters. The name may include alphanumeric characters (A-Z, 0-9) and underline. Space is in other words not allowed.

The slides in a block group are saved in the file SLIDES.SLB in the block group library.

Every block group has a database designation connected to it.

You can work with several MLUSER trees at the same time (See chapter “Structure and definitions”).

Blocks

A block is placed in a symbol library. To each block you can attach an icon (slide). In each library you can place any number of blocks. Every block has a database designation connected to it. The database designations include information about how the block shall be set up and descriptive information (to help choose a block).

The database designations for the block group are copied when you create a new block. This means that a block, which belongs to a certain block group library, automatically receives the group's basic properties.

The block name must be a valid file name and must not include more than 32 characters. The name may include alphanumeric characters (A-Z, 0-9) and underline. Space is in other words not allowed.

Icons

An icon (slide) can if desired represent each block. This icon is used as support when you choose which block in a library that should be inserted. You can also choose if you want to show the description of the block. The functions, which create icons automatically, insert a cross at the insertion point of the block on those icons that are created.

Network support

The block group directories are designed to be used by several users in a network. To avoid destroying other users data, a block is locked so that other users cannot work with the block at the same time.

Registration of external blocks

If you have blocks that have been created without BLOCKLIB, these can easily be registered in a symbol library.

Procedure:

1. Create a block group with the command in BLOCKLIB. You can also use an existing group (it may contain other blocks).
2. Copy the blocks and potential accompanying slides (i.e. SLD files with the same name as the blocks) to the library with the help of the operating system's commands.
3. Use the function **Read**. The library is registered and the new blocks added in at the bottom of the list of blocks in the library.

If you want, you can also create an icon for each block with the function **Slides.** Note that this erases all objects on the current drawing (including the configuration).

How to create a symbol-report

There are two methods to print symbols, one fast and easy and one custom method to suit your own individual needs.

The Standard method

This method is usually used at two occasions. First, when you quickly want to review the contents of a symbol group and maybe change one or more symbols. Secondly, when you want to distribute symbols between users. This is done by "importing" the symbols format and data about the symbols (descriptions etc.) so that it is stored in the same location in a drawing.

Procedure:

1. Create a new drawing.
2. In the pull-down menu, select **Blockbuild; Blocklib**.
3. Select **Admin**.
4. In the dialog box which appears, mark the group COILS (IEC113) and wait until *el*Master lists the symbols in the window.
5. Press **Report**<.

6. Make sure that **Standard** is selected and that the symbols do not “explode” when they appear in the drawing.
7. Select **OK**.

*el*Master will place the symbols in the group of the drawing. All that remains is for you to print out and save.

The Custom method

With this method, reports are created from BLOCKLIB, where the correct appearance of a report is important. This requires some preparation. First a pre-defined drawing format must have been incorporated in the drawing. You use this format to describe what the report will look like and also which information and blocks will be shown. The procedure for this method is similar to the Standard report procedure, the difference being that you must first start by inserting/incorporating a drawing format.

Procedure:

1. Create a new drawing.
2. Insert a new drawing form intended for Custom Blocklib-reports.
3. In the pull-down menu, select **Blockbuild; Blocklib**.
4. Select **Admin**
8. In the dialog box which appears, mark the group COILS (IEC113) and wait until *el*Master lists the symbols in the window.
5. Press **Report<.**
6. Make sure that **Custom** has been selected and give the report a name. This name will be used for the drawing files, which have been created by the function.
7. Select **OK**.

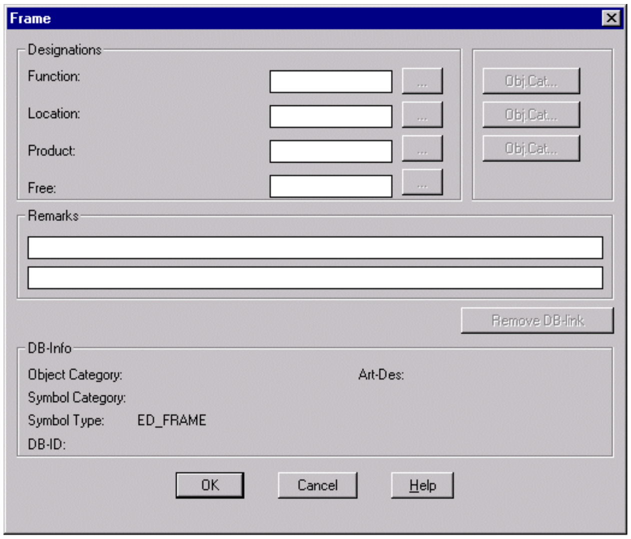
*el*Master will place the symbols in groups, on the drawing. If there is not sufficient room on the page for all symbols, this function will save the drawing in a drawing file and at the same time create a new drawing for the remaining symbols. The drawings are saved in the current directory. The file name is built up of the name you gave the report and the page number (i.e. COILS1.DWG, COILS2.DWG and so forth).

Item designations

Item designations are used in the drawing to show if respective post in the plant's structure diagram is function- or location orientated.

The dialog box for entering item designations looks different depending on which type of object that has been chosen. There is one general dialog box for objects, and a dialog box specially suited for frames, signals and cross-reference symbols.

The dialog below, Figure 29, is used for boundary frames.



Symbol Type: ED_FRAME

Figure 29. Frame

The dialog, for handling designations, use three different designation fields. These are:

- *Block 1* (labeled *Function*) with prefix "=": tells **what** is done. The equal sign is used in the drawing to show that the block is a function designation.
- *Block 2* (labeled *Location*) with prefix "+": tells **where** it's done. The plus sign is used in the drawing to show that the block is a location designation.

- *Block 3* (labeled *Product*) with prefix "-": tells **who** is doing it. The minus sign is used in the drawing to show that the block is a product designation.

If the drawing is connected to project database, you have the possibility to look-up existing objects in the database with the help of the button with three dots.

Designations can be configured to use either IEC 750 or the new standard IEC 1346 (NEWIEC). The main difference between these standards is how the different designation types are concatenated, and how they form a complete designation.

To be able to handle the different standards it is important to know which rules that apply on each object in the drawing. In this section we have illustrated some examples to show different cases that can occur. For more information about the different standards see respective standard:

- IEC 750, Rules for electrical diagrams – Item designation
- IEC 1346-1, Structuring principles and reference designations – Basic rules

Boundary frames are handled in the same way by both standards and is described in next section.

Designation structures – Standard IEC 750

The standard IEC 750 defines two structures of designations. These are:

- Function designation
- Location designations

These structures can be used one at the time, or both at the same time. It is recommended to combine blocks 1 and block 3 when you want to derive unique item designations and block 2 when you want supplementary information on the location.

Function designation

The function designation consists of block 1 and block 3. Designations of block 1 are always at a higher level than block 3. Block 3 can never be at the topmost level.

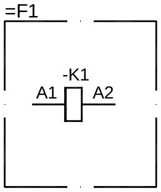


Figure 30

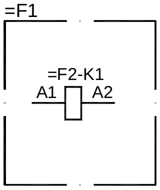


Figure 31

Figure 30 and Figure 31 shows two different ways of using block 1 and block 3 correctly.

The complete function designation in Figure 30 is =F1-K1 and in Figure 31 =F1=F2-K1. Note that the local designation for the coil in Figure 31 has been entered as a designation block 3 (Product).

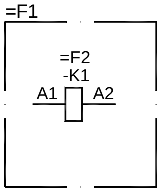


Figure 32

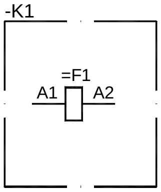


Figure 33

Figure 32 and Figure 33 show incorrect ways of using block 1 and block 3. The reason that it is incorrect in Figure 32 is that block 1 and 3 are at the same level and in Figure 33 that block 3 precedes block 1.

Location designation

This designation consists of designation type block 2. If designation block 2 is missing, but a designation block 3 exists, this will be used instead.

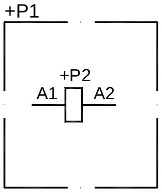


Figure 34

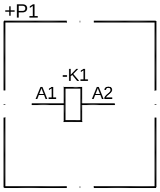


Figure 35

The complete location designation in Figure 34 is +P1+P2 and in Figure 35 +P1-K1.

Absolute designation

When the location designations for the symbols within the frame are not to be concatenated with the sheet-common item designations or outlying frame the prefix > can be used. You can also use the double-dotted frame.

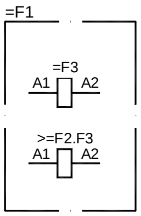


Figure 36

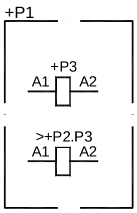


Figure 37

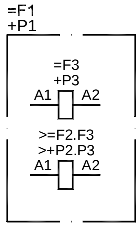


Figure 38

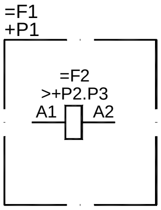


Figure 39

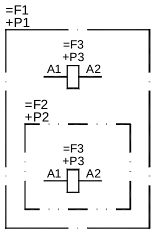


Figure 40

	Complete function designation	Complete location designation
Figure 36	=F1=F3 and =F2=F3	
Figure 37		+P1+P3 and +P2+P3
Figure 38	=F1=F3 and =F2=F3	+P1+P3 and +P2+P3
Figure 39	=F1=F2	+P2+P3
Figure 40	=F1=F3 and =F2=F3	+P1+P3 and +P2+P3

Upper designation

Normally the designation shown at a symbol is the local designation for the symbol. In some situations, it is needed to show an upper designation at the symbol. If this is done, the checkbox “Upper” must be marked. This checkbox gives information to the analyze-function not to create a local designation for the object. See Figure 41 and Figure 42 for example.

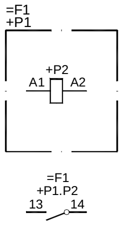
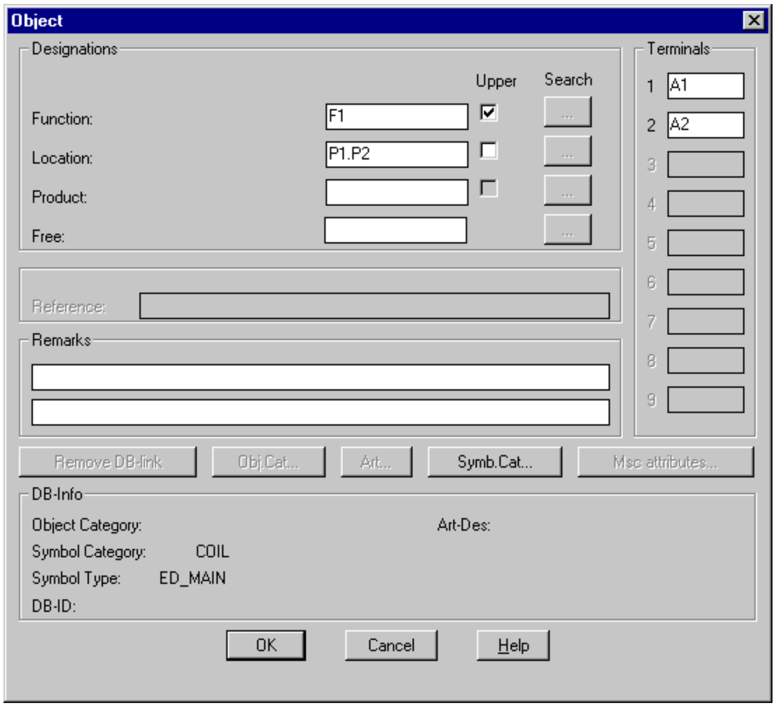


Figure 41



Function: F1 ☑

Location: P1.P2 ☐

Product: ☐

Terminals: 1 A1, 2 A2

Object Category:

Art-Des:

Symbol Category: COIL

Symbol Type: ED_MAIN

DB-ID:

Figure 42

In this example, both symbols will give the same designations. In the bottom symbol, the checkbox “Upper” has been used to indicate that the designation =F1 is an upper designation for the symbol.

Transition within a designation block

It is possible to use transition within each designation block. The prefix of the type with which it is transitioned must be entered, see Figure 43 and Figure 44.

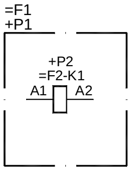
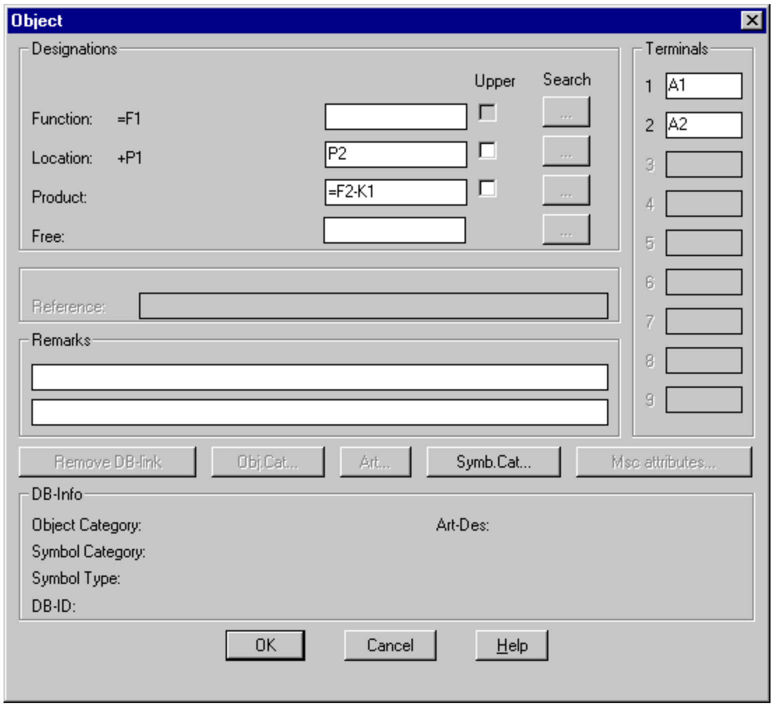


Figure 43



Object

Designations

			Upper	Search
Function:	=F1		☐	...
Location:	+P1	P2	☐	...
Product:		=F2-K1	☐	...
Free:				...

Terminals

1 A1
2 A2

Reference:

Remarks

DB-Info

Object Category:
Symbol Category:
Symbol Type:
DB-ID:
Art-Des:

Figure 44

Below follows two other examples of transition within a designation block.

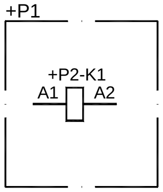


Figure 45

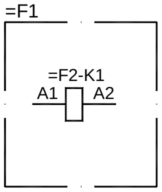


Figure 46

	Complete function designation	Complete location designation
Figure 43	=F1=F2-K1	+P1+P2
Figure 45		+P1+P2-K1
Figure 46	=F1=F2-K1	

Examples of correct combinations

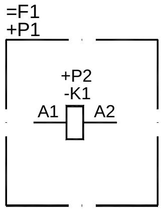


Figure 47

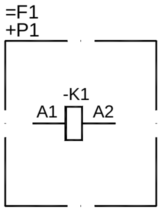


Figure 48

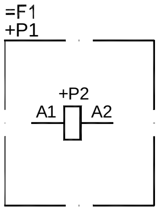


Figure 49

	Complete function designation	Complete location designation
Figure 47	=F1-K1	+P1+P2
Figure 48	=F1-K1	+P1
Figure 49	=F1	+P1+P2

Examples of incorrect combinations

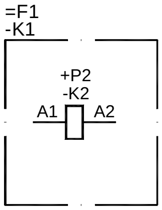


Figure 50

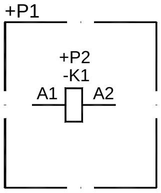


Figure 51

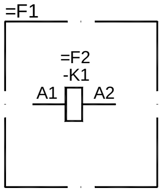


Figure 52

	Error
Figure 50	Block 3 (-K1) as topmost designation. Block 1 is missing
Figure 51	Block 3 (-K1) as topmost designation. Block 1 is missing
Figure 52	Block 1 (=F2) and block 3 (-K1) at same level

Designation structures – Standard IEC 1346

The standard IEC 1346 defines tree structures of designations. These are:

- Function structure
- Product structure
- Location structure

Concatenations in each structure

Each structure is treated individually, and forms its own structure.

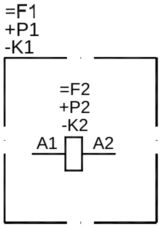


Figure 53

	Complete function designation	Complete location designation	Complete product designation
Figure 53	=F1=F2	+P1+P2	-K1-K2

Transitions between structures

To create a transition from one structure to another, the prefix of the upper designation is added to the prefix of the lower.

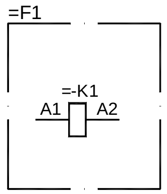


Figure 54

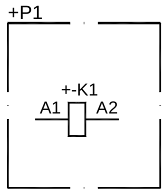


Figure 55

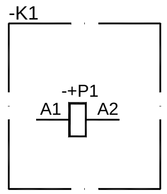


Figure 56

	Complete function designation	Complete location designation	Complete product designation
Figure 54	=F1-K1		
Figure 55		+P1-K1	
Figure 56			-K1+P1

Absolute designation

The same rules apply to IEC1346 as for IEC 750.

Upper designation

The same rules apply to IEC1346 as for IEC 750.

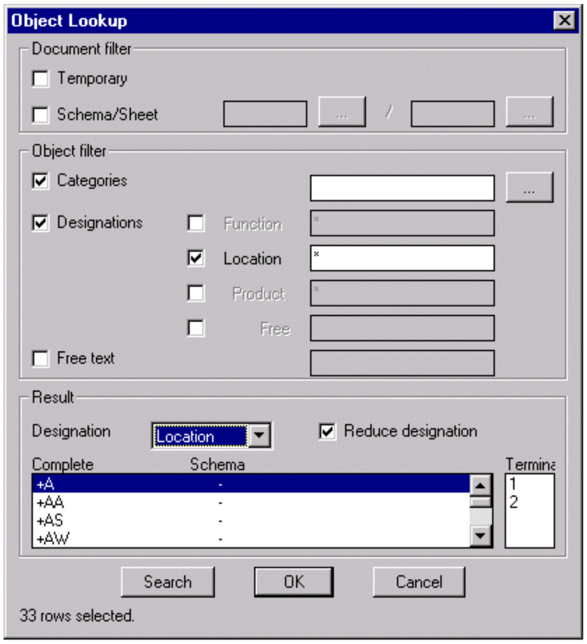
Transition within a designation block

The same rules apply to IEC1346 as for IEC 750.

Look-up and linking to an object

To search for a specific level or unit in a plant for example a cubicle, a function or an apparatus, the *Object dialog* is used. This dialog will show if you press the button with three dots in the dialog for inserting item designations.

To be able to effectively search in large amounts of data different filters can be used.



Object Lookup

Document filter

☐ Temporary

☐ Schema/Sheet

Object filter

☑ Categories

☑ Designations

☐ Function: *

☑ Location: *

☐ Product: *

☐ Free

☐ Free text

Result

Designation: Location

☑ Reduce designation

Complete	Schema
+A	-
+AA	-
+AS	-
+AW	-

Termina
1
2

33 rows selected.

Figure 57. Object look up

Note: This is not basic functionality in *el*Master design. *el*Master project is also required.

Document filter

The document filter is used to filter out designations that are used in the drawing and in the database. The function **Temporary** presents the apparatuses that are not to be found on any sheet. The function **Schema/Sheet** presents the apparatuses that are found in a diagram. It is also possible to specify the designation and sheet number at the search.

Object filter

The object filter is used to search for "objects" of a specific kind (category) or a specific designation. If the symbol, which is to be inserted, has an upper designation in form of a boundary frame or a common designation for the sheet, this topmost designation will be shown as default when doing the search.

Reducing the upper level designation in the search result

The complete designation is stored in the database. When working in a circuit diagram using boundary frames, and where part of the object's designation is found, the designation at the symbol should only show the last level of the designation. To reduce the designation, use the function **Reduce designation**. The filled in search value is used to reduce the designation (automatic filled in if upper level can be found).

Boundary frames

Boundary frames are used in a drawing to indicate items that belong together, physically, mechanically or functionally.

There are three different types of boundary frames. The difference between them is the line-type with which they are drawn.

- The frame with the single-dot-dash line is normally used for specifying details of a device in accordance with IEC standards.
- The boundary frame with the double-dot-dash line is used when the location designations for the symbols within the frame are not to be combined with the sheet-common item designations or outlying frames.
- The continuous frame does not follow a standard.

The complete designation

When you are working with a so-called detached designation system and are using boundary frames to group a number of symbols, which have common designations, only the part of the designation that not is found in the frame should be written in the symbol. If you, for example, have an apparatus with the location designation +H.1.D1.20, a drawn frame for a cubicle with the designation +H.1 and inside that a frame with the designation +D1, the designation for the apparatus should be set to +20. The complete upper part of the designation is shown in the dialog in front of the editing fields. The object's complete designation can be settled by adding the part in front of the field with the part in the field.

For signal designations, the upper level is also shown. What type of designation that will be used as a prefix is defined in the configuration that the drawing applies.

Connection lines

There are three different functions for creating connections. These are:

- **Free connection line**

  Is the base function for manually creating a connection line, every corner point must be selected.

- **One connection line**

  With this function only start and end point have to be selected. The connection line continues in the same direction as the connection point of a symbol or the end point of a connection line.

- **Multi connection line**

  Creates a number of parallel connection lines with a fixed distance between start and end point.

All connection lines can be repeated, i.e. the function can be repeated by pressing ENTER. By pressing ENTER at point selection gives the last point specified. This feature can be used to make several connections consecutively.

Automatic line break-up

All connection lines can be drawn over existing symbols. Automatic line break-up will occur at the points that are defined as terminals or where there is a node (**POINT**). Lines are also joined and made continuous.

Figure 58 shows a coil with four nodes (each node is illustrated with a cross). Two lines have been draw across the coil with the function **One conn.line.** As you can se in the picture automatic line break-up has occurred at all four nodes.

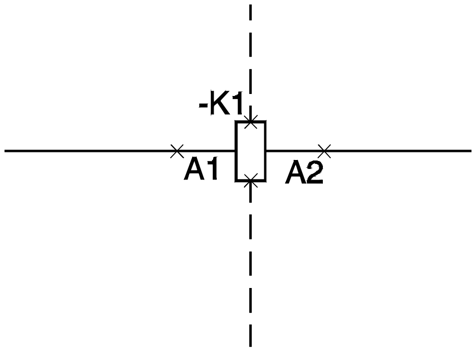


Figure 58. A coil with four nodes (POINTS)

For automatic line break-up to occur the line has to mark the node exactly. To make this happen as smoothly as possible it is an advantage to have SNAP (AutoCAD's command) on. The SNAP function turns on an invisible grid and snaps the screen crosshairs and all input coordinates to the nearest point in the grid. The spacing between points of the SNAP grid is adjusted in the configuration dialog.

When specifying a point, the function OSNAP (Object Snap) provides a sight that automatically finds the correct connection point (searches for a connection point or another conductor). Which connection point the function should search for is specified with the function **Object Snap** in the **Assist**-menu.

Examples on how to draw connection lines

In Figure 59 the function **One connection line** has been used, start and end point are illustrated with a cross. Which point you select first does not matter.

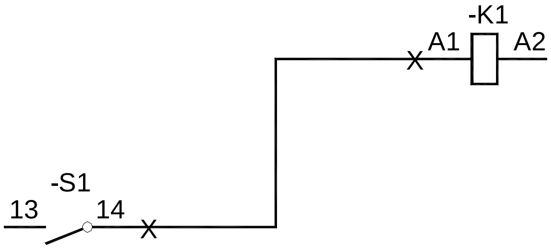


Figure 59

The function **One connection line** can not always choose the right direction. When the start point is situated in an intersection with several connections, the function can not automatically choose in which direction the new line should be drawn. The dotted lines in Figure 60 show the different ways the connection line could take. In this case, we recommend the function **Free connection line** to enable a correct path.

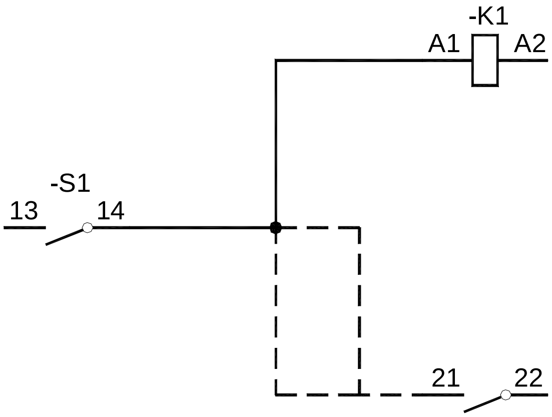


Figure 60

If the start point or the end point is positioned on a connection line the function will insert a connection point as shown in Figure 60. A single connection point can be created by entering a start point on a line or intersection and then finishing the command by pressing ENTER.

When the start point and the end point are both free points (not on a connection line or connection point), the first line will be drawn in the X-direction from the start point. In the examples below (Figure 61), the start and end point are illustrated with a cross and selected in the order as shown.

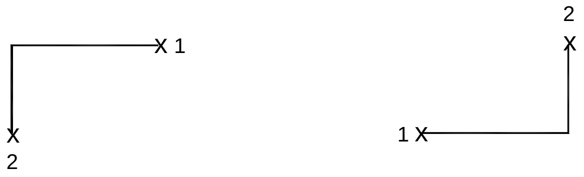


Figure 61

References

References are used the same way as *Connection lines* to link conductors, but instead of connection lines a value, a signal designation, is given to the connections that are being linked. References are used, for example, when you document an electrical installation or a plant where the circuit diagram is separated on a large amount of sheets.

In most cases cross-references are together with references. These make it possible to follow a circuit within a sheet or between different sheets. Read more about cross-references in the next section.

Note that references are not for deciding in which order a circuit should be connected. References are created so that you may find the next occurrence of a signal or a supply in the circuit diagram.

There are three types of references, *general*, *source* and *sink*. A general reference gives you cross-references in a chain from the first appearance to the last. Source and sink references create radial cross-references, which have their roots in the sink reference. Read more about cross-references in the next section.

References should always be inserted at the end point of a line. Special rules apply for the scope of a reference (the same rules apply for location designations of symbols). If a reference is going to apply to a complete or part of an installation depends on how you specify the value for the reference. In Figure 62 the signal “M-“ will apply to the complete installation. The signal “ON” will apply to the unit “+J1”. The signal “OFF” refers to sheet “123” therefore, an equivalent signal should be found on this sheet.

Compared to the earlier versions of *el*Master, the equivalent signal to ”M-“ was “supply”, and the equivalent to “ON” was “signal”, and the equivalent to “OFF” was “schematic/sheet”.

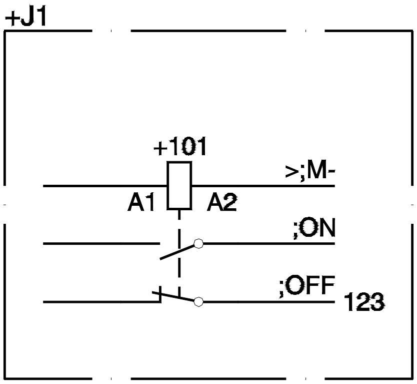


Figure 62. A symbol with references

The sign “;” is in accordance with IEC 750 and is a prefix for a signal designation, and the sign “+” indicates the location designation.

Cross-references

Cross-references specify where further information within the sheet or within the same document can be found. The information can consist of the drawing number (document identity), sheet number or column and row within the sheet according to the drawing form’s reference net.

A plant can have a number of apparatuses with symbols that are spread out over several sheets. Between these apparatuses there are also connections that can run over many sheets.

To which degree the apparatus's symbols are spread out depends mainly on two things. The first thing is the type of drawing that you are creating, for example if the drawing is created using a detached or attached representation. The second thing is the orientation of the drawing, for example if it is location or function.

In drawings that have been created with a detached representation cross-references are usually needed between apparatuses symbols as the symbols can be drawn on different sheets.

In drawings that have been created with an attached representation it is not so common with cross-references between apparatus symbols. On the other hand cross-references between signals and supplies are more common.

Remark: If you have access to *el*Master project you don't have to specify the cross-references as the program does this automatically!

Cross-references between the symbols of apparatuses

Necessary components (see also Figure 63):

- A main symbol. It is usually the symbol that corresponds to the maneuver instrument of the apparatus, e.g. the coil for a relay.
- One or more sub symbols, e.g. the contacts for a relay.
- Cross-reference symbols, which are placed next to the main symbols and are used to display certain references.

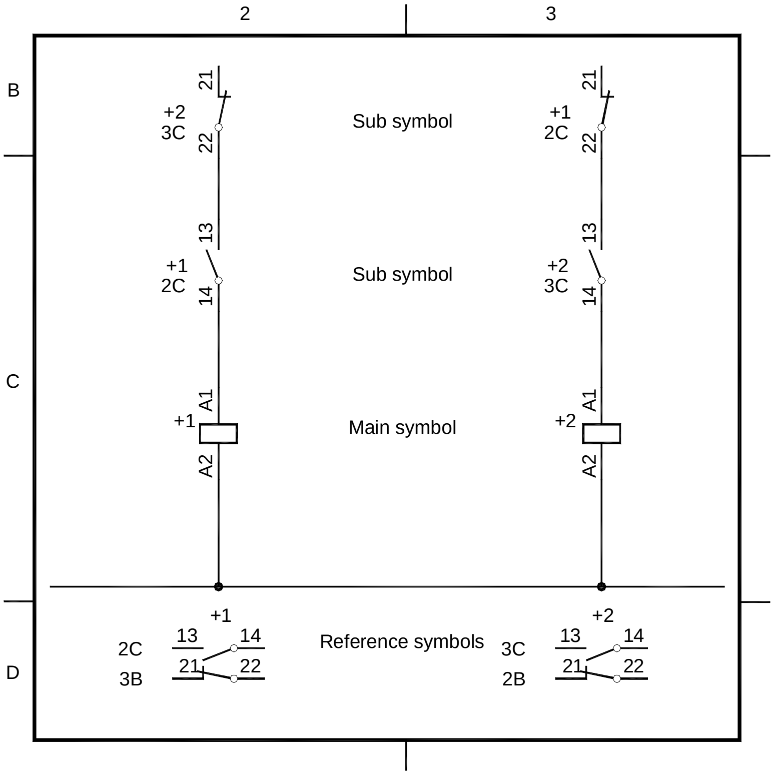


Figure 63

It is important that you use the right symbol type when you insert the symbols. The main symbols should have the type ED_MAIN and the sub symbols any type except ED_MAIN. The cross-reference symbol should have the symbol type ED_XBLK.

Cross-reference symbols will automatically be inserted if they are not inserted manually. The symbol that is inserted automatically is a table in which designations and references are shown.

If you like a reference symbol with a specific appearance you have to insert it manually. You can choose which cross-reference symbol you like to insert from the **Symbols**-menu. There are a number of cross-reference symbols to choose between.

When you insert the cross-reference symbol you have to connect it with the main symbol that it belongs to. The connection is necessary for the function, which creates and updates the reference, to find the right symbol. The connection is made either when the symbol is inserted with the help of the button that is found in the dialog box for cross-references or afterwards with the function **Connect Symbols** (se the Command Reference).

Note that some of the cross-reference symbols that are delivered with the product are compound symbols and will therefore not give a dialog box. For these symbols the connection is done afterwards. In order for the function, that handles the connection, to work the main symbols must be of the right type, i.e. ED_MAIN. Also respective sub symbol and belonging reference symbol must have the same symbol category.

It is not necessary to specify the reference symbol's item- and terminal designation. The function that creates and updates the reference will overwrite these values. However, terminal designations that are filled out and that correspond to the terminal designations that exist in any sub symbol will be used to direct the update function to the right symbol.

Remark: How to create your own cross-references is described on page 123

Cross-references for signals and supplies

The type of reference decides the nature of the cross-reference (general, source or sink).

General reference

A general reference gives you cross-references in a chain from the first appearance to the last. The function first selects references within the sheet, from left to right and from top to bottom, then references between different sheets.

References on the same sheet and in the same net are ignored, so that no cross-references are created between them.

Source-Sink references

Source and sink references create radial cross-references, which have their roots in the sink reference.

When you use source-sink references you should remember that there can be only one sink reference. You should also remember to allow space for the reference texts.

Cables and Wires

In *el*Master design you can specify cables and wires both in draft and with design data. Cables and wires specified with design data are used by connection table programs to create connection tables.

Draft

It is possible to graphically specify cables according to IEC 617-3. However, it is only a “draft” description i.e. the information will not be used by the connection table program. To create a “draft” description, use the command **Spec.Cond.** which you will find in the **Symbol**-menu. From here you can, for example, choose between different “draft” symbols to insert on the connection and specify the number of wires in a connection.

With the function **Cable** under the **Conn/Ref.**-menu you can also describe the cores in a connection.

Design data

Note: the following text and examples are adapted for *el*Master interconnect to create connection tables.

It is possible already in the drawing to decide which type of cable/wire that should be used to connect two points. It is also possible to show a cable/wire that is of special interest.

In an electrical diagram the exact connections are usually not drawn, but only graphical descriptions of which terminals that are connected. If an electrical point consist of more then two terminals, the electrical point can be constructed in several different ways. To decide or show cables in an electrical diagram the sewing order for the electrical point must be specified, as you only want one way to construct the electrical point.

The function that is used for drawing a cable/wire is called **Design data (Interconnect)**. With this function you can also assign a connection a new or an existing cable/wire.

Restrictions in the drawing

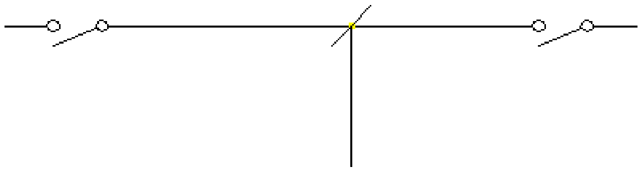
It is not possible to remove or change an existing connection in a drawing that is stored in the database.

If a marker is incorrect i.e. it is not clear which connection that is intended, the marker will be ignored when saving or updating from database.

Below follows six examples of correct and incorrect cable markers.

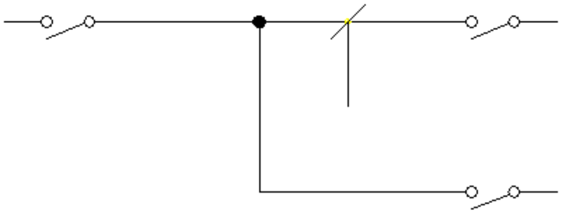
Note that the information about the cable or wire data will be updated when the electrical diagram is updated from database. If the connection is specified from the drawing, the drawing has to be saved to database for the cable or wire information to be updated when updated from database.

Example 1



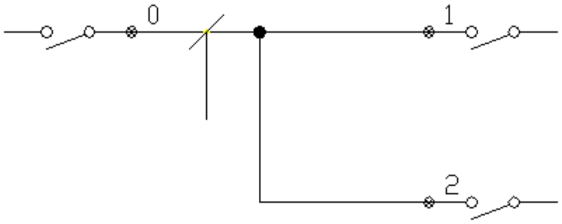
Correct marker: This electrical point is the easiest, with only two terminals. No extra information is necessary, as there is only one way to construct it.

Example 2



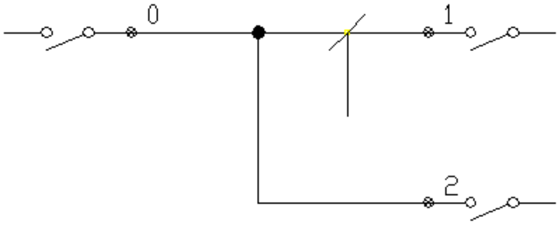
Incorrect marker: This electrical point has more than two terminals which means that it is impossible to decide which connection is meant.

Example 3



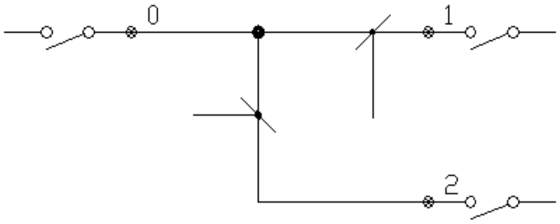
Correct marker: In this electrical point the sewing order is specified. As the order is decided it is possible to decide which connection that is meant, in this case the connection between terminal 0 and 1.

Example 4



Incorrect marker: In this electrical point the sewing order is specified but there still exist an ambiguity. It is not possible to decide if it is the connection between 1 to 0 or 1 to 2 that is meant.

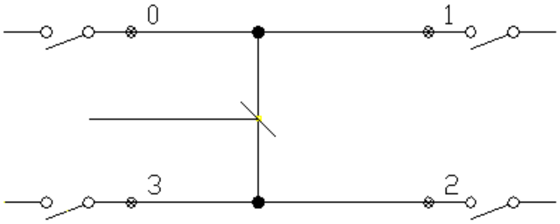
Example 5



Correct marker: In this example the ambiguity in example 4 is solved by also specifying the connection between terminal 2 and 1.

The new marker specifies a connection between terminal 2 and another terminal with the sewing order number 2 ± 1. In this case, it is only one terminal that satisfy these criteria, terminal 1. Since the new marker specifies the connection between terminal 2 and 1, the other can only specify the connection criteria between 1 and 0.

Example 6



Incorrect marker: In this example the marker is on a line that is not directly connected to a terminal. The marker will therefore be ignored.

Dimensioning

The dialog for the Dimensioning command follows AutoCAD's dialog for dimensioning as far as possible. Since the standard is not completely covered with AutoCAD's dimensioning, there are some commands that have an adjusted dialog. Among these are superimposed running dimensioning and two different types of diameter dimensioning. Common to all dimensioning types is the input of dimension text and placing of the dimension line.

Dimensions can be done according to the standard ISO 129.

Dimension line placement

The placement of the dimension line for linear dimensions can be performed in four different ways:

- **Press ENTER**

  If you press ENTER when you are specifying the placement of a dimension line, a pre-set distance from the object is selected. This pre-set value can be changed during configuring.

  The function asks on which side of the object the dimension line shall be placed. You can specify this by pointing at the required side.

- **Specify a distance from the object**

  When the dimension line is to be placed, a screen menu is shown with the following alternatives:

  8
  10
  12
  Free
  Relative

  8, 10, 12 give a distance of 8, 10 and 12 mm respectively from the object. Select desired distance (8,10 or 12) from the screen menu. **Free** prompts for a distance (arbitrary distance can be specified).

  The function asks on which side of the object the dimension line should be set. Pointing at the desired side specifies this.

- **Specify a distance from a separately given point (Relative)**

  Select **Relative** from the screen menu.

The function prompts for a distance from a relative point. It is possible to choose 8, 10 or 12 from the screen menu, enter the desired distance (mm) or choose a distance by specifying two points.

The function then prompts for a relative point. It is this point the earlier given distance should originate from.

- **Specify a point the dimension line should pass through.**

  Select the point the dimension line (or its extension) should pass through.

Input of dimension text

The dimension's measure value is displayed with ten decimal points. The appearance of the dimension text in the drawing is shown below. Choose from the following alternatives:

- **Press ENTER.**

  Accept the proposed dimension text or choose from the alternatives shown below.

- **Write in a new dimension text.**

  If this is done, the dimension text will not be updated if the measure is STRETCHED.

- **PADD or PSUB from the screen menu.**

  Increase or decrease the number of decimals in the dimension text.

- **PREFIX and/or SUFFIX from the screen menu**

  Allows input of prefix and/or suffix to the dimension text.

Dimension texts

AutoCAD interprets text dimensions in a particular way.

- An empty text (ENTER only) is interpreted as if the measure value should be displayed instead.
- The character combination “<>” is interpreted as if these characters ought to be replaced by the measure value. This can occur in the middle of a text.

In both cases, the measure value follows if the measure is STRETCHED. If a measure value with numbers is entered manually, the measure value will be CONSTANT.

Example: The text “123.45 mm” is constant whether or not the measure is STRETCHED or of how long the measure actually is.

The text “<> mm” will be displayed as “12.34 mm” if the measure is that long, but will change accordingly as the measure is STRETCHED.

Editing

Under the **Edit**-menu there are several functions for editing objects. Some of the functions listed here invoke AutoCAD standard functions. These are described in the AutoCAD manual.

When you edit in *el*Master design you first give the command and then select the object you want to edit. You can either select the object with the pointing device or with a command under **Select Objects** in the **Assist**-menu. See the AutoCAD-manual for more information about these commands.

Grips

Changes to connection lines and frames are easier to do if **grips** is switched on. To move a corner point, mark the line and the corner to be moved. Pull the corner to the required position. Note that it is also possible to move, rotate, copy and do various other operations to the selected object.

Refer to the AutoCAD manual for more detailed information about “grips”.

Ed_Copy, Ed_Move and Ed_Erase

This function is not the same as AutoCAD's **COPY**, **MOVE** and **ERASE**.

Copying, moving and deleting is subject to the same rules as with the insertion of a symbol on a line or when a line is drawn over a symbol. Even lines, which are broken during a branch-off, are joined again when they are copied moved or deleted. However in some occasions, the function may not be able to re-join a line. This may occur when a symbol is moved beyond the connection line. It may also occur if a symbol is moved and causes one of the terminals to be placed where a line was previously broken. If either of these occur, use the function **STRETCH** instead.

Global changes

Using this function as support, it is simple to make the same changes that are common for a number of different drawings.

The different kinds of changes that the function supports are the following:

- Do a PURGE (see the AutoCAD-manual) of a block at several levels.
- Changing values of attributes in blocks. Example: Changing text in the title block of the drawing frame, or changing item designations in a circuit diagram.
- Changing text strings. Can for example be used if one wants to translate simple describing texts.
- Changing the title block and transfer attribute texts which is very helpful when a customer wants his own title block on the drawings.
- Exchanging blocks and transfer attribute texts.
- Free function, defined in LISP.

Configuration

You can make certain settings to influence the program's manner of working. These settings are stored in each drawing. The settings can be changed with the configuration function in the program. You can export configuration information from the drawing to a file that can later be used to change the settings of another drawing.

Basic configuration

The function **Config**. opens a dialog box where you can make configurations in *el*Master design. The following headings are shown:

- Format
- Layer
- Attribute handling
- *el*Master Diagram
- *el*Master Schema
- Dimensioning
- Item list
- Import
- Export

You can read more about these functions in the Command Reference on page 194. In appendix B, “Configurations”, you will also find the variables that control respective part.

Scale factor

All parts of a drawing that have an equivalent in reality should be drawn on a scale of 1:1.

To fit large objects within a given format, you enlarge the drawing frame a number of times until everything fits. Later, when the drawing is plotted, shrink the drawing (both objects and drawing frame) to the normal size.

This technique has many advantages. If you draw all objects in a 1:1 these can without any trouble be moved from drawing to drawing. For the parts that should have a certain size on the final paper copy of the drawing, texts, dimensioning, etc. must be adjusted to the chosen scale factor.

When you select scale factor in the application, set how much, all parts that is to have a certain size on the paper, should be scaled up. It is difficult to change the scale factor after the drawing is completed. It is relatively easy to re-scale the drawing frame, but many of the other parts that are dependent on both the paper and the objects (for example dimensioning) are difficult to scale properly.

Storing the configuration information in the drawing

The configuration information is stored at a point in the drawing origin on the layer SOS_DOCCONFIG. If the configuration point is removed from the drawing the settings that have been made disappear.

The layer SOS_DOCCONFIG is normally locked and frozen when you work on a drawing so that you cannot remove or change the configuration by mistake. The program unlocks the layer when the configuration is changed and then automatically locks the layer again.

Storage of configuration information outside a drawing

You can store the configuration in a file with the extension CFG. This file can then be registered (imported) to the configuration of other drawings.

Note: The construction of the configuration information was inspired by AutoCAD's attribute. Each value that can be configured has a label (or identifier) and a value. For example: the label of the revision marking block is REVBLOCK and the value "revring".

Automatic updating of the configuration

When you start editing a drawing, the application searches for a file called SETTING.CFG in the same directory as the drawing. If this file is found the application updates the drawing's configuration information with the settings that exist in SETTING.CFG. SETTING.CFG is created by exporting the configuration information from a drawing.

In this way you can make a simple configuration that can be used for all drawings in a project.

New drawing

When you start up a new drawing or edit a drawing, which was created outside *el*Master design, the program inserts a configuration into the drawing with pre set values. The application then searches for the file SETTINGS.CFG in the drawing's directory. If this doesn't exist, the program searches for the file DEFAULT.CFG in the directory CFG of the program tree. The drawing's configuration is updated with information in some of these files.

Note: In the same manner the application searches for the file LAYERS.CFG (in the directory of the drawing and, if it doesn't exist, it searches in the CFG in program code tree). LAYERS.CFG contains of the layer structure description (see description of layer system).

Plotting

AutoCAD has the possibility of configuring many different plotters, which you can switch between (see the AutoCAD manual). To be able to separate the different plotter configurations, we recommend to give each plotter configuration a description. This description is used by *el*Master's PLOT function to set the correct plotter configuration. Note that this implies that AutoCAD has been correctly installed.

The directory for plotter settings is found in the MLUSER-tree. The MLUSER-tree consists of one or more PCP files and a file with the description of the plotter configuration (plotter configuration name). The PCP-file contains settings describing the procedure of the plotting.

If you want to plot to a file, the file name should be selected in the PCP file.

Scale factor

You can either plot with SCALE or with FIT. If you want to plot in SCALE, the drawing is set to the scale factor it has been configured for. If you plot with FIT the drawing becomes as large as possible according to the plotter used and will therefore NOT be properly scaled.

In each directory containing plotter settings, a number of PCP files are stored that contains settings describing the procedure of the plotting.

If you decide to plot in SCALE there must be a PCP file in the directory for plotter settings with the same name as the internal name of the format type. Please notice the distinction between the name of the format type (e.g. A3 Landscape, A4 Page) and the internal name of the format type (e.g. A3L, A4P). Otherwise the function is aborted.

If you decide to plot with scale factor FIT the function uses the PCP file that has the same name as the internal name of the format type. If this is missing, you are asked to choose a new PCP file among the PCP files existing in the directory.

LIBDATA.TXT

The file LIBDATA.TXT contains the plotter's description as well as a description line for the directory. There is a LIBDATA.TXT for each plotter that is configured for the system. The files can be found in the corresponding plotter library under \MLUSER\PLOT.

FORMTYPE.CFG

The file FORMTYPE.CFG contains the connection between the name of the form type and the internal name of the form type. It is an ordinary ASCII file and it is placed in the directory CFG under MLB.

Network support

The functions handling the drawing forms are written to function in a network environment in which several users share the same settings. The libraries are locked when they are being used so that a user not should be able to modify the contents while another user is working with the library.

Example

Make the following settings:

1. Configure AutoCAD for the plotter to be used. Enter the plotter description.
2. Create a directory for plotter settings. Save the plotter configuration description in the directory's setting.
3. Use AutoCAD's Plot command to save a PCP file in the directory for plotter settings. The PCP file should have the same name as the internal name of the format type (for example, A31.PCP for the format type A3L).

Note: The setting of scale factor and the setting on HIDE in the PCP files are superseded by the current values when plotting.

During plotting:

1. Register a format type in the configuration by inserting a form. Format type can also be registered without the application's function for insertion of forms if used.
2. Decide whether or not the plotting should include hidden lines (HIDE).
3. Select scale factor (SCALE or FIT) and which part of drawing to plot (LIMITS, EXTENTS etc.).
4. Select which directory with plotter settings should be used.

The following occurs:

- The function instructs AutoCAD to work with the plotter with the description registered in the directory with plotter settings.
- The function searches for the PCP file in the directory with plotter settings that have the same name as the format type. If such a file is missing and you have chosen to plot with the scale factor FIT you will instead need to choose among the existing PCP files in the directory. If the PCP file is missing and you have decided to plot with scale factor SCALE the function is aborted.
- If the correct PCP file exists or you have decided to use another PCP file, its settings are registered. Note that the setting of scale factor and HIDE in the PCP file is changed to the current values. The settings of these values in the PCP files will thereby be changed.
- The plotting is started with the selected settings.

Prototype drawing

A prototype drawing is a drawing used as a pattern when you create a new drawing (see the AutoCAD manual for a description of how AutoCAD handles prototype drawings)

You can work with prototype drawings in two different ways:

- Prototype drawing with configuration.
- Prototype drawing without configuration.

Prototype drawing with configuration

This is the easiest and fastest way to work with a prototype drawing. This method is used if you want to have the same layer system for all drawings.

Work procedure:

1. Start AutoCAD with a drawing without configuration.
2. Start the application. The configuration is now loading and the configuration system is being created. Specify a scale factor of 0.0.
3. Set the AutoCAD variables as required.
4. Define the text styles and font files to be used.
5. Save the drawing.
6. Configure AutoCAD to use this drawing as a prototype drawing (see The AutoCAD manual).

When a new drawing is chosen, the following occurs:

- AutoCAD makes a copy of the prototype drawing.
- The application reads the configuration in the drawing (including the configuration for the layer system). If there is a configuration file with the name SETTINGS.CFG in the same directory as the drawing, the drawing will be updated with its settings.
- Ready for drawing.

Prototype drawing without configuration

This is the most flexible manner of working with prototype drawings. This method is used if you want to have different layer systems on different drawings. You can, for example, have different collections of layer groups (but with the same base group!) on different drawings.

By not having the definition of the layer system set in the base drawing, the definition is now registered from a file when you start up a new drawing.

To create a drawing without configurations, you must start AutoCAD without starting the application (cancel with Ctrl-C at an early stage).

Work procedure:

1. Start AutoCAD with a drawing without configuration. Note that you should start AutoCAD without starting the application.
2. Set the AutoCAD variables as required.
3. Define the text styles and font files to be used. Select the menu file to be used.
4. Save the drawing.
5. Configure AutoCAD to use this drawing as a prototype drawing (see The AutoCAD manual).

When a new drawing is chosen, the following occurs:

- AutoCAD makes a copy of the prototype drawing.
- The application confirms that the configuration is missing and registers this (see configuration). The application searches first for the file SETTINGS.CFG in the drawing's directory, and if it does not exist, it searches for DEFAULT.CFG in the program tree's MSC directory.
- The application confirms that the configuration of the layer system is missing and registers this (see layer system). The application searches first for the file LAYERS.CFG in the drawing's directory and if it does not exist, it searches for LAYERS.CFG in the program tree's MSC-directory. If the layers in the configuration are missing, these will be created.
- Ready for drawing.

elMaster EPS

CHAPTER 5

elMaster EPS is a part in elMaster design. EPS analyzes diagrams and produces data used by elMaster InnerFT/Cable/connect to produce connection tables.

Procedure

The procedure for EPS-generation is to first generate **Net/sheet** (EPB) for every sheet and then **Net/diagram** (EPS) for all sheets. The result of the EPS-generation is kept in a text file, which is used by the various connection table programs. Information from the EPS-generation is saved in a database. If you want to remove a sheet, which you do not want to keep in the EPS-file, you can use the function **Elp/delete** (EPR). If the EPS program reports any errors or warnings, you can use the functions **Show error**, **Error list/sheet** and **Error list/diagram**.

System description

When you start a project, a directory EPSDB is created in the same place as the drawing files. The files in EPSDB are copied from the directory ELMASTER/DIAGRAM/TAB/DESEMPDB. If you want to do a back up of connection information about your project, then all files in EPSDB must be saved. During execution there will be a lock file (DESLOCK) placed in EPSDB. If any of the EPS-programs are ended by mistake, then the file DESLOCK will still remain. To be able to execute the program again, it will be necessary to delete this file.

It is possible to configure the EPS-program from the files EPSLIST.CFG and BLADERR.CFG. Both of these are in the EPSDB-directory.

Error reports from EPB- and EPS-generation

Included in *el*Master EPS is an advanced in-built error checking function. The purpose of this is to help the user in the work of analyzing the circuit diagrams. The messages can be displayed in AutoCAD’s text window.

```
=======================================================
====================================================
Sheet found
====================================================
----------------------------------------------
Diagram: -
Sheet : -
----------------------------------------------
ERR-290 Electrical point with free connections x:277.50 y:200.00
====================================================
Electrical points per sheet
Errors: 1
Warnings: 0
====================================================
```

There are two types of error: Warnings and Errors. Warnings do not affect the operation of the program. The next stage of the processing can be done in spite of the warning. If an error occurs, it must be corrected, and a new EPB analysis must be done to verify the diagram.

These errors are stored in the files EPSERR.TXT respectively EPBERR.TXT. These files are stored in the directory EPSDB under the relevant project. The file shows which diagram, which sheet and from where the error originates. Every error is then specified, line-by-line, with the error code, a description of the error and the position in the drawing.

Note that certain errors do not have a position; these are given the co-ordinates X: 0.00 Y: 0.00.

Finally, an end-of-report message appears together with the total number of Errors and Warnings on the sheet.

Error codes

The table below lists the error codes with explanations and suggestions for how to correct them. The error type is also given using “E” for error and “W” for warning.

Code	Type	Reason	Action
100	E	Not enough disc space	Delete unnecessary files
110	E	Error reading file	Check file
120	E	Cannot open file	Check if file exist, if it does, contact system manager
140	E	Database engaged	Check if another user is operating EPS. If not, delete the file DESLOCK in the EPSDB-directory
150	E	Cannot find database	Path to database is probably too long
160	E	Configuration error	The standard, separator or prefix is missing from the configuration
170	E	Empty database	You are using EPS on an empty database
180	E	Too low status	There are sheets in the database which have not been correctly EPB processed
200	W	A free Net attribute has been found	Edit the drawing
210	W	There are two identical Net attributes for the same Net	Move or delete one of the attributes
220	W	A free terminal attribute has been found	Edit the drawing
230	W	There are two identical terminal attributes in the same terminal	Edit the drawing
240	E	A free Net end has been found	Edit the drawing
250	W	There are two symbols with item designations on the same mechanical link	Edit the item designations the sheet

Code	Type	Reason	Action
260	W,E	Boundary frame not closed	Edit the frame on the sheet
270	E	More than 40 characters in item designation	Reduce the number of characters
280	W	Two main symbols on a mechanical link	There can be an item designation selection problem
290	W	No value for the signal	Enter the value
300	W,E	Record is missing	Check the error message and try to correct, if this fails, contact the system manager
310	W	Attribute has incorrect format. Does not agree with the configuration in BLADERR.CFG	Edit attribute
320	W	Attribute value is missing. Does not agree with the configuration in BLADERR.CFG	Enter the attribute
330	W	Reference between sheets is not correct	Edit reference in sheet
340	E	Free reference	The reference can have been inserted on a “draft” line. Edit the line
350	W	Double attribute	Two attributes of the same type on the Net or a terminal
360	W,E	Cannot find terminal	There are more terminal nodes than attributes. Can be an error in the symbol. Even reports symbols which are to be used to connect to a mechanical link
400	E	Too many boundary frames on sheet	Limit to maximum 50
410	E	Too many line segments on sheet	Limit to maximum 1000
420	E	Too many symbols on mechanical link	Limit to maximum 100
430	E	The boundary frame has too many line segments	Limit to maximum 20
440	E	Too many symbols within the boundary frame	Limit to maximum 250
450	E	Too many symbols on sheet	Limit to maximum 500
460	E	Too many characters at designation level	Number of characters/levels limited to 10
500	E	The UPD-file has no equivalent DWG-file	Remove the UPD-file
510	E	Two units have the same number	Edit the drawing
520	E	Cannot find a unit	The unit may of been removed from the drawing

Code	Type	Reason	Action
600	E	Error in link file	Problem with data in the link file
610	E	Error in drawing's database	Serious AutoCAD-fault
620	W	Designation does not follow standard	Edit the designation
630	E	Problem with DCL-file	Error in the DCL-file
900	E	Internal database error	Correct the other errors on the sheet. If the error still remains, contact the system manager

The EPS-file's appearance

After successful EPS-generation, the result is stored in a text file (ASCII-file). The content of the file describes how the nets of the circuit diagram are linked.

A default file name is offered, based on the immediately superior product name, followed by the file type EPS, as follows: <<project name>>.eps

Since the EPS-file is open and readable, the contents can be used by all uses to create tables.

What is written in the file is governed via a configuration file EPSLIST.CFG. More about the configuring of this and other settings are given in "The file EPSLIST.CFG" on page 109.

The EPS-file starts by list of all the terminal blocks in the circuit diagram. If this listing is not required, the variable "!TB" in EPSLIST.CFG must be set to "N". The nets of the circuit diagram follow the terminal blocks, one by one. Each net, in turn, is broken down into connection points.

Code	Meaning
!TB	New terminal block begins
!BL	Sheet where terminal block is located.
!TI	Terminal block designation (PPB)

Code	Meaning
!TN	Terminal (Number/side)
ORD	Order name
*AD	Sub-project
#EP	New net begins
KSH	Name of circuit diagram
ELP	Net number, picked from reference, if available. To get the definite signal designation, add SIG to "E" in EPSLIST.CFG
+AP	New connection point begins
FPB	Function-oriented item designation
PRO	Product-oriented item designation
PPB	Location-oriented item designation
OBJ	Symbol type, e.g. OPEN_CONTACT, I/O-CHANNEL
SYM	Internal symbol type i.e.: ED_MAIN, ED_XREF
ABS	Definite address, applies to I/O-symbols
SYA	Symbolic address, applies to I/O-symbols
PLT	Indicates that the connection point is a terminal block.
UTG	Terminal number
BLD	Circuit diagram sheet
XPT	X-co-ordinate
YPT	Y-co-ordinate

In addition to the codes in the table above, the EPS file may contain further information linked to nets or symbols, for example, voltage class or type of apparatus. This information is controlled by the variables EPP and KPP in the file EPSLIST.CFG.

The file EPSLIST.CFG

This configuration file specifies what information is to be written to the EPS-file when **Nets by diagram** is called.

The file begins with a numeral, which specifies the number of configuration lines that follow. Each configuration line consists of a variable name followed by “E” or “N”. The variable names correspond to codes in the EPS-file. An “E” at a variable means that the codes corresponding to this must be printed out and an “N” means that they must not be printed out.

The variables correspond to codes as shown in the table below:

Variable	Corresponding code(s)	Initial config.
!TB	!TB, !BL, !TI, !TN	E
ORD	ORD	E
*AD	*AD	E
#EP	#EP	E
KSH	KSH	E
SIG	SIG	N
EPP	Net attribute	E
+AP	+AP	E
DPB	DPB	N
FPB	FPB	E
PRO	PRO	N
PPB	PPB	E
OBJ	OBJ	N
SYM	SYM	N
ABS	ABS	N
SYA	SYA	N
PLT	PLT	E
UTG	UTG	E
BLD	BLD	E

Variable	Corresponding code(s)	Initial config.
KRD	XPT, YPT	N
KPP	Component attribute	E

The file BLADERR.CFG

Specifies which block attributes are to be checked. The attribute tag and the format of the attribute values are checked.

The attributes that can be checked are:

- Component attribute
- Net attribute
- Terminal designation

These attributes and their values can be checked for:

- Existence
- Format

Number of configuration lines (excluding first line).

<Attribute type> <Existence check> [<Attribute tag>] <Format>

...

...

Attribute type

Three types can be specified:

K = Component attribute

N = Net attribute

U = Connection attribute (Attribute to terminal designation)

Existence check

Can be set on “E” or off “N”.

Attribute tag

This refers to the tag of the attribute in AutoCAD. Can only be specified for component attributes and Net attributes.

Component attributes that can be specified:

ID = Location-oriented record designation.

IDF = Function-oriented record designation.

Net attributes that can be specified:

ELP = Net number

SPK = Voltage class

ARE = Area

SPC = Specification

CBL = Cable to which net belongs

PRT = Core type

Format

The format specifies the structure of an attribute value. Four characters are used in the specification:

A = ASCII, all letters A - Z.

D = Digit, all numerals 0 - 9.

.= Delimiter, approved delimiters apart from point are +, - and =.

X = All characters except delimiters.

Example of a format specification: ADDD.AXXX

Here, an approved attribute value must start with a delimiter, followed by no more than one ASCII character, no more than three digits, one delimiter, no more than one ASCII character and finally, no more than three characters other than delimiters.

So, the format specification lays down an upper limit on the permitted number of characters. This rule applies to A-, D- and X-types, but not for the delimiter type, which must be as specified.

An approved value with the above specification might look like this: .K1.A2

Restrictions

Below follows a table, which shows the restrictions that exist in *el*Master EPS.

Number of characters in document identity	22
Number of characters in sheet number	10
Total number of characters in designation	80
Number of characters in designation per level	20
Number of characters in terminal and signal reference	20
Number of terminals per symbol	9
Number of characters in sheet reference	22
Number of characters in net data	20
Number of characters in connection point data	20
Number of characters in remarks	40
Number of characters in OBJECTTYPE	20
Number of characters in SYMBOLTYPE	20
Number of characters in path to EPS-database	40

Create your own block

CHAPTER 6

elMaster contain a large amount of symbols. A symbol is a block, which includes part of a circuit diagram.

This chapter describes how you create your own blocks.

Block builder menu

*el*Master design contains a well-designed function for building symbols. You will find the function in the pull-down menu **Blockbuild**.

To make you understand how **Blockbuild** works, we will begin with a short exercise. We will build the symbol shown in Figure 64.

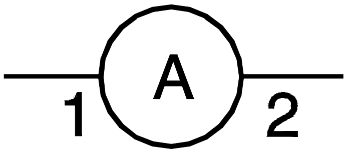
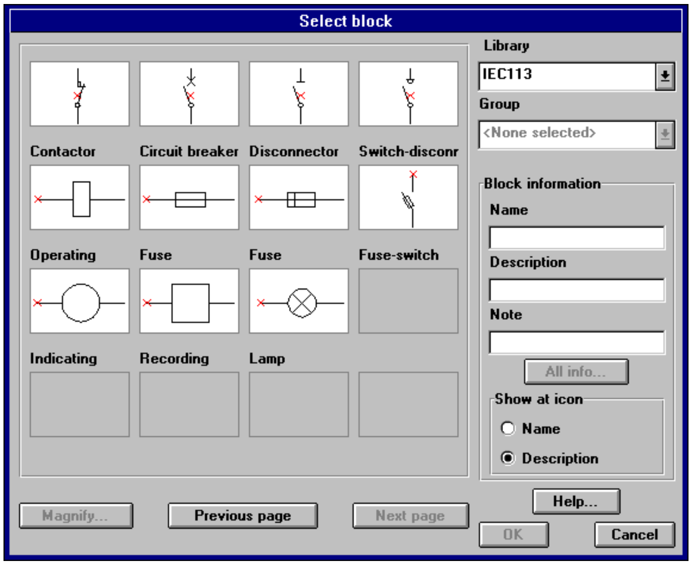


Figure 64

Primitive

The symbol shall have the properties to enable us to change the text in it, i.e. we shall be able to specify which type of instrument the object in question is. The illustration above shows a voltmeter.

1. Select **File:New** to create a new drawing
2. Select **Blockbuild**; **Initialize**.
3. Select **Blockbuild**; **Primitive**. The dialog box shown in Figure 65 will be displayed.



Library: IEC113

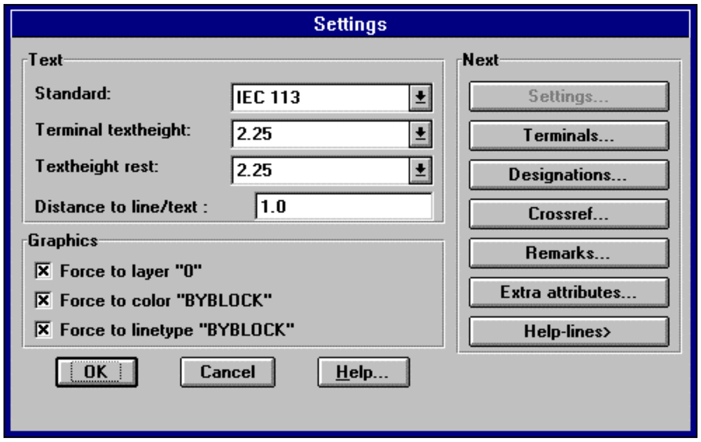
Group: <None selected>

Figure 65. Select block

This is where you will find the so-called primitives, for example base symbols, which accord to standard.

4. Choose the symbol **p080101h** and press **OK**.
5. Select **Blockbuild**; **Settings**. The dialog box in Figure 66 will show.

Settings



Text

Standard: IEC 113

Terminal textheight: 2.25

Textheight rest: 2.25

Distance to line/text : 1.0

Graphics

☒ Force to layer "0"

☒ Force to color "BYBLOCK"

☒ Force to linetype "BYBLOCK"

Figure 66. Settings

6. Specify the values as given in Figure 66.

You have now chosen the standard IEC 113, which means that the terminal designations will be placed below the connection lines. The text height of the terminals and other symbols is 2.25 mm. That the value is 2.25 and not 2.5 is because of the thickness of the text, which normally is 0.25 mm. The distance between the connection line and the top part of the terminal designation is 1 mm.

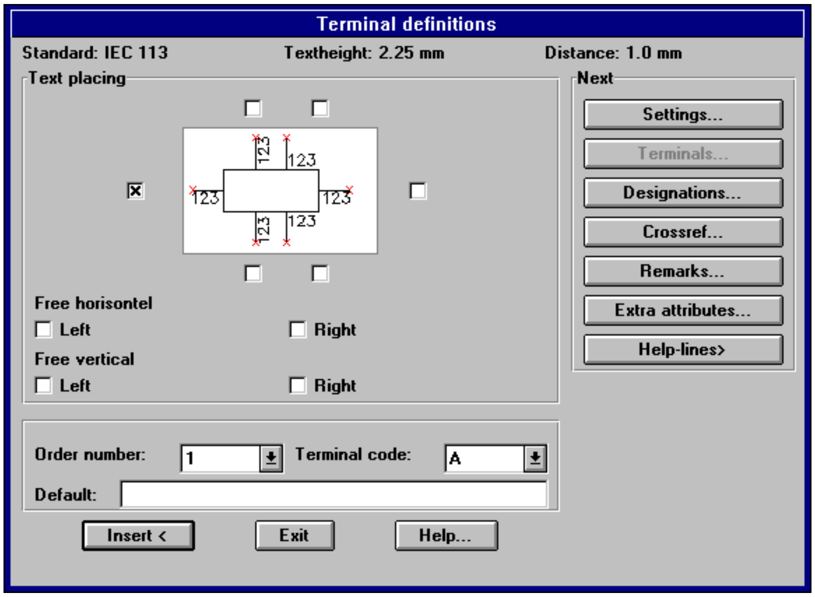
With the alternative **Force to layer “0”** you choose if the symbol should be placed in layer 0 and if it will acquire the properties to choose color and line type according to the layer it is found in. This will only occur if colors and line type are set to **Bylayer** in your drawing.

To the right in the dialog box, you can see the title **Next** and under this title there are several alternatives. These are also available in the pull-down menu, but by selecting them in this dialog box you avoid having to go to and from the different functions.

7. Select **Terminals**. The dialog box in Figure 67 will show.

Terminal

The dialog box shown in Figure 67 is used to show where the terminal in your symbol should be. The work procedure is that you select the alternative you want for each terminal and then select **Insert**. Do this again for all the terminal definitions.



Standard: IEC 113
Textheight: 2.25 mm
Distance: 1.0 mm
Order number: 1
Terminal code: A
Default:

Figure 67. Terminal definitions

8. At the top of the box, you can find some of the earlier settings you have done. Make sure that these are correct.

Under the title **Text positioning**, you can set the position of the text to your terminal in relation to the point you will give later. If you set the correct type here, you will automatically get the correct position of the terminal. You have access to six locked and four free alternatives. Usually, you will only use the locked alternatives, as in this example.

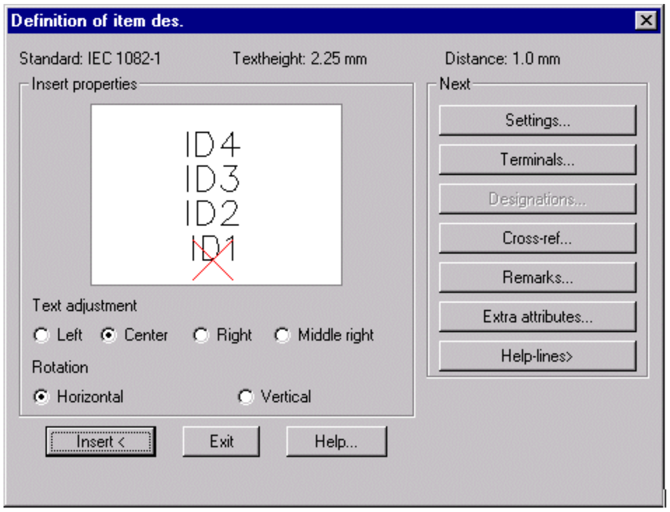
There are also titles called **Order number**, **Terminal code** and **Default value**. The Order number is the data which *el*Master uses, for example, to make a connection table. It is not necessary for you to know all about these titles, but remember these points:

- Each value must only apply to one terminal in your symbol. For example, it is not possible to have more than one terminal with the designation 1A.
- The *el*Master automatically counts the values each time you create a terminal designation.
- When you create the terminal designation, you will see that for each Order number there is a “mate”. Do not change this.
- The default value is a pre-set value, which you get as a recommendation when you insert the complete symbol.

9. Make the settings, as shown in Figure 67, to insert the left-hand terminal of the symbol.
10. Point at the **Insert** command and mark the position of the left-hand terminal (at the end of the "stick").
11. Align the text with the right hand terminal. Make sure the order number counts one step and give a default value of 2.
12. Select **Location designation**. The dialog box in Figure 68 will show.

Location designation

We shall now insert the attribute definitions for the symbol’s location designation.



Definition of item des.

Standard: IEC 1082-1
Textheight: 2.25 mm
Distance: 1.0 mm

Insert properties

Text adjustment
○ Left ● Center ○ Right ○ Middle right

Rotation
● Horizontal ○ Vertical

Figure 68. Definition of item designation

You can choose between four text adjustments. By pointing at the various alternatives, you can see what the location designations will look like. If you want to, you can rotate them 90 degrees.

13. Select **Center** and **Horizontal** and then **Insert**.
14. *el*Master will now prompt you for the insertion point. To achieve the exact position, enter **Qua** on you keyboard and press ENTER.

    Qua is an abbreviation of the word Quadrant and is one of many “object snaps" in AutoCAD. **Qua** searches for one of the circle's four quadrants (the one nearest to where you point will be chosen).
15. Point at the uppermost of the circle. The symbol will now look like shown in Figure 69.

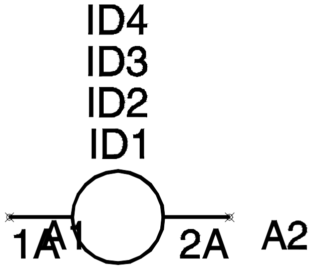


Figure 69

16. Select **Remark**.

Remarks

The alternatives for the remarks are similar to those of the location designation.

17. Select **Center** and **Horizontal** and then **Insert**.
18. Enter **Qua** on you keyboard and press ENTER.
19. Point at the lowest point of the circle. The symbol will now look like Figure 70.

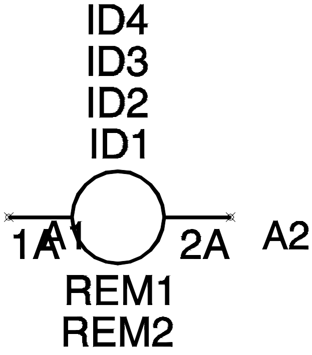


Figure 70

20. Select **Exit** in the dialog box.

Cross-reference

You have probably noticed that we have not used all the alternatives available. For example, we did not select **Cross-reference**. The reason for this is that cross-references are not used in this type of symbol. They are used with symbols, which are linked to others, for example a contact or a coil. If these are drawn on different sheets, you can use the cross-reference to specify where the contact or coil is.

Extra attribute

There is also an alternative called **Extra attribute**. You can use this function to add further attribute definitions to the symbol. By marking the required definition, you can choose what you want to add.

Point

Point is AutoCAD’s point command. Sometimes it can be an advantage to insert points in a symbol. If you will find a specific point in a symbol, you can use the object snap **Node**.

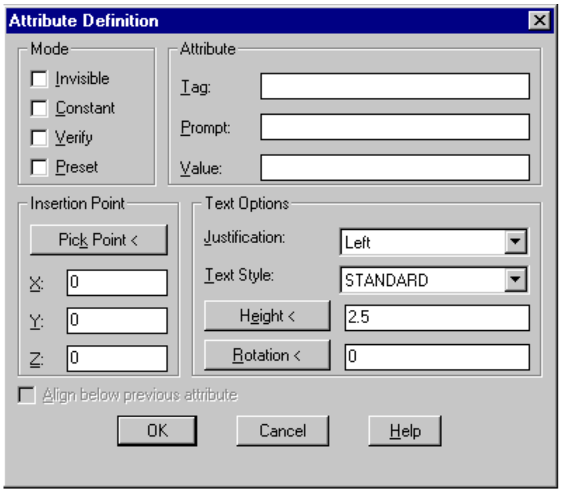
Help-lines

Help-lines give you extra help when you want to insert an attribute under or over an existing one. Select **Help-line** and mark the earlier attribute, one or more. *el*Master will mark the attribute with a frame, which you can use as a reference point (for example by selecting **Midpoint**) during insertion of the next attribute definition.

DDATTDEF

DDATTDEF is AutoCAD’s command to create attribute definitions. Use this command if one of the earlier definitions is not suitable. In our symbol we shall use DDATTDEF to insert the last attribute (the attribute that will contain the instrument type).

21. Select **Blockbuild; DDATTDEF**.
22. Fill in the dialog box as in Figure 71 and then select **Pick point**.



☐ Invisible
☐ Constant
☐ Verify
☐ Preset
Tag:
Prompt:
Value:
X: 0
Y: 0
Z: 0
Justification: Left
Text Style: STANDARD
Height < 2.5
Rotation < 0
☐ Align below previous attribute

Figure 71. Attribute definition

23. Mark the circle's center. Use the object snap **Center**. (Enter **Center** in the prompt and then press ENTER.
24. Select **OK** in the dialog box. The symbol should now look like Figure 72 and is now ready to make a block of.

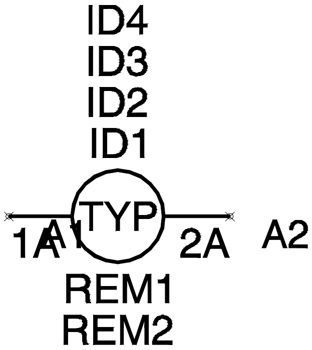


Figure 72

25. Select **Blockbuild; Blocklib; Admin**. A dialog box will show, which handles the placement of symbols in symbol libraries.

Se “Blocklib” on page 145 for more information of how you handle the dialog box *Block administration*.

Cross-reference symbols

Cross-reference symbols are symbols that are used to make a reference between the different symbol parts of an apparatus. The cross-reference symbols usually look the same (graphically) as the symbol you want to refer to. In *el*Master the difference between a symbol and its cross-reference symbol is that the cross-reference symbol has attributes for references instead of terminal nodes.

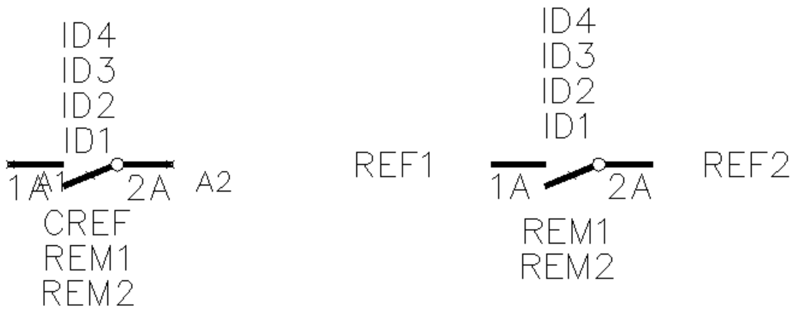


Figure 73

Figure 73 shows a symbol definition for a make contact and its cross-reference symbol. Notice how the terminal nodes in the reference symbol have been removed.

Layout symbols

There are many different ways to create blocks for layout drawings. A layout drawing in its simplest form consists of cubicles, groups and apparatuses drawn in the shapes of circles and squares. With the function **Free symbol** you can create these drawings without having to create any special symbols. If you want to create a more realistic drawing, you will need to create symbols for the articles that you are going to use.

Own symbols

There are many symbols already created in the program, but because of the large amount of articles, a complete system that fits everyone, is impossible to create. But these symbols can be used as examples when you are creating your own symbols. These symbols have been created while taking maximum advantage of the different functions available in the drawing environment, for example layers and levels (z-co-ordinate).

The symbol management in *el*Master makes it possible to define the characteristics that are used when inserting a symbol. This makes it possible to define several different types of symbols. You can, for example define a symbol that will explode when inserting it. This is useful when you want to create combined symbols, i.e. symbols that consist of other symbols. You can also decide where you want to place the symbol by giving an insertion point. This can be useful when you want, for example an inserted terminal on a bar to hide the part of the bar that is behind it. You can do this by setting the insertion point of the terminal to x,y,6 and the insertion point of the bar to x,y,4. If a symbol is a bar and you have to define its length, you do this by specifying the symbol type to ED_RAIL (See also “Symbol types” page 281).

When you are creating your own symbols it is important that you follow certain standardization about the appearance and shape of the symbol. The symbols that come with this product have been created using the following rules (see also Figure 74):

- Scale 1:1.
- The outer contours have been drawn with layer 01. The other lines in the symbol have been drawn with layer 02. Center lines have been drawn on layer 05. Center lines should always be included, because they are very helpful when you are inserting and dimensioning.

- When the mounting area is the same as the outer contours, they are created with AutoCAD's 3DFACE command on layer 04. 3DFACE is used so that you can remove hidden lines with the HIDE command.
- On certain front mounted apparatuses (e.g. push buttons) the part of the apparatus that is behind (inside) the mounting area is larger than the rest of the apparatus. This is drawn as a mounting area, with the 3DFACE command, on layer 04 and on a level that is behind the rest of the symbol.

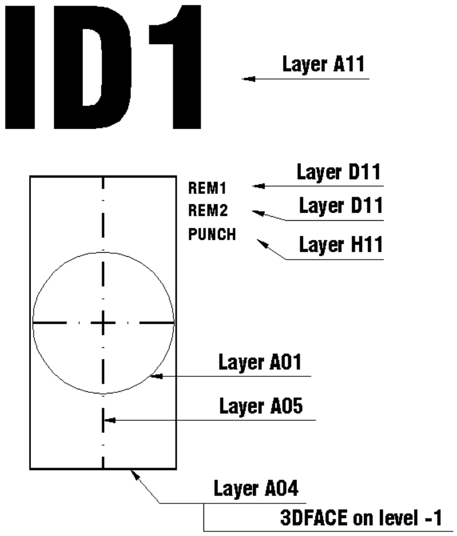


Figure 74

Layer/layer groups for symbols:

Type	Part of Symbol	Remark	Layer
Enclosure/ cubicle	Outline	C= Cubicle/Cabinet	C01,C02
	Mounting area		C04
	Center line		C05
	Dimension, line		C08
	Dimension, text		C12
	Attribute, id	Local oriented item des.	C11

Type	Part of Symbol	Remark	Layer
	Attribute, other		D11
Group	Outline	P=Panel/Plate	P01,P02
	Mounting area		P04
	Center line		P05
	Dimension, line		P08
	Dimension, text		P12
	Attribute, id	Local oriented item des.	P11
	Attribute, other		D11
Apparatus	Outline	A=Apparatus	A01,A02
	Mounting area		A04
	Center line		A05
	Dimension, line		A08
	Dimension, text		A12
	Attribute, id	Local oriented item des.	A11
	Attribute, other		D11
	Attribute, hole punch ref.		H11
Hole punch		H=Hole	H01,H02,H03
Label/Sign	Outline	L=Label	L01,L02
	Attribute, id	Local oriented item des.	L11
	Attribute, text		L11
	Attribute, other		D11
Design Aid		D=Design	D01-D16

Conventions for naming symbols

This section describes the rules that apply to naming symbols. One of the main problems when you make symbols is to find names for them. This section describes how you can name your symbols, so that they are easy to manage. The fundamental demand for this convention is:

- Maximum eight characters (in accordance with. DOS).
- Numeric and alphabetic characters only (UNIX does not accept the $ character for example).
- Give basic information about the symbol, for example, the type of drawing it should be used for, type of symbol, etc.
- Easy to name similar symbols.

The name is composed of a number of codes, where the first character describes what type of diagram or drawing the symbol is designed for.

The first character codes:

K = Circuit diagram

L = Layout drawing

E = Single line diagram

B = Block diagram.

To tell what type of apparatus the symbol represents, the codes for function-oriented item designation, block 3, in IEC750 is used.

Supplier can be indicated in those cases where the symbol not is general, for example for circuit diagram symbols not in standard and layout symbols.

To explain the specification in the simplest way we have divided it into the different type of drawings and symbol-types.

Layout-drawing

Supplier specific symbols (enclosures, instruments, contactors etc.)
L+Apparatus type+Supplier+Code+ Variant
Apparatus type: One letter in accordance with IEC 750.
Supplier: One letter (see “Supplier codes” page 129).
Code: Four characters. Part of the order number, extension number or other similar number.
Variant: One letter (see “Variant Codes” page 129).

Standard symbols (Bars, sockets etc.)
L+Code+Variant

Code: Six characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129

Circuit diagram

Symbols according to IEC617

K+Apparatus type+IEC-number+Variant

Apparatus type: One letter in accordance with IEC 750.

IEC-number: Five digits (leading zero in IEC-number removed).

Variant: One letter (see “Variant Codes” page 129).

Combined symbols

K+Apparatus type+Code+Variant

Apparatus type: One letter in accordance with IEC 750.

Code: Five characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129).

Supplier specific symbols

K+Apparatus type+Supplier+Code+Variant

Apparatus type: One letter in accordance with IEC 750.

Supplier: One letter (see “Supplier codes” page 129).

Code: Four characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129).

Single line diagram

Symbols according to IEC617

E+IEC-number+Variant

IEC-number: Six digits.

Variant: One letter (see “Variant Codes” page 129).

Combined symbols

E+Apparatus type+code+Variant

Apparatus type: One letter according to IEC 750

Code: Five characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129).

Supplier specific symbols

E+Apparatus type+Supplier+Code+Variant

Apparatus type: One letter according to IEC 750

Supplier: One letter (see “Supplier codes” page 129).

Code: Four characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129).

Block diagram

Symbol according to IEC617

B+IEC-number+Variant

IEC-number: Six digits.

Variant: One letter (see “Variant Codes” page 129).

Combined symbols

B+Apparatus type+Code+Variant

Apparatus type: One letter according to IEC 750.

Code: Five characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129).

Supplier specific symbols

B+Apparatus type+Supplier+Code+Variant

Apparatus type: One letter according to IEC 750.

Supplier: One letter (see “Supplier codes” page 129).

Code: Five characters. Part of the order number, extension number or other similar number.

Variant: One letter (see “Variant Codes” page 129).

Qualifying symbols

For example symbols that can be used in different types of diagrams and drawings.

According to IEC617-2
Q+IEC-number+Variant
IEC-number: Six digits.
Variant: One letter (see “Variant Codes” page 129).

Other
Q+Code+Variant
Apparatus type: One letter according to IEC 750.
Code: Six characters. Extension number, code etc.
Variant: One letter (see “Variant Codes” page 129).

Application specific symbols (references etc.)

For example symbols specific for certain application, for example references, blocks for codes etc.

A+Code+Variant
Code: Six characters.
Variant: One letter (see “Variant Codes” page 129).

Supplier codes

One letter according to the following:

A = ABB
T = Telemecanique
S = Siemens
Etc.

Variant Codes

One letter according to the following:

A = Variant “A”

B = Variant “B”

C = Variant “C”
H = Horizontal
V = Vertical
L = Left
R = Right
U = Up
D = Down
F = Front

Applied examples

Layout symbol for contactor type ABB EH40:
LQAEH40U.DWG
L = Layout
Q = Acc to IEC 750 switch for main circuit
A = Supplier ABB.EH40=Code.U=Variant Up

Circuit diagram symbol for contactor type ABB EH40:
KQAEH40H.DWG
K = Circuit diagram.
Q = Acc to IEC 750 switch for main circuit
A = Supplier ABB.
EH40 = Code.
H = Variant horizontal.

Layout symbol (frontal drawing) for enclosure type VSH 700:
LAAVSH7F.DWG
L = Layout
A = Acc to IEC 750 function unit
A = Supplier ABB
VSH7 = Code
F = Front

Circuit diagram symbol NO-contact (IEC 617-7 no 07-02-02):
K070202H.DWG

K = Circuit diagram
070202 = IEC-number
H = Variant horizontal

Circuit diagram symbol for signal-lamp with serial resistor:
KH12345H.DWG
K = Circuit diagram.
H = Acc to IEC 750 signal equipment
12345 = Code.
H = Variant horizontal

Structure of Directories

CHAPTER 7

The applications, included in elMaster Power Concept, are designed to function well together. In order to enable sharing data and set-ups the applications should be installed in the Common-directory.

In this chapter the structure of the dictionary is described.

Structure of Directories in *el*Master design

*el*Master design is one of the applications included in *el*Master Power Concept and is normally installed with the other applications in the *el*Master -directory. Files that are common for the applications in *el*Master Power Concept are found in the Common-directory.

The files that are specific for *el*Master design you can find in the design-directory. The files that can be found in the *el*Master-directory are divided into a number of sub directories, these are described in the table below. As *el*Master design is started with AutoCAD there is no EXE-file in the directory.

Directory	Description
BDB	Directory for the block database
CFG	Configuration directory for drawing settings.
ELCFG	Configuration directory for settings of AutoCAD and elMaster, does not need to be found in the design-directory.

Directory	Description
IEC 113	Directory with symbol library for the standard IEC 113.
IEC1082	Directory with symbol library for the standard IEC 1082.
LAYUSER	Directory with symbol library for layout drawings.
MLUSER	Directory for own made symbols
MSC	Program files and other help files
TAB	Template for creating EPS database

Symbol Libraries

The symbol libraries contain all the symbols that exist in *el*Master. You are free to modify, change or delete files from these. However, it is important that any such modifications are well documented so you will be able to recreate these when updating the application.

You choose which symbol library you want to use by selecting **MLCONFIG; set ini** in the screen menu. You can specify a number of search path by separating them with a semicolon. The settings are stored in the file MASTLINE.INI.

The symbol library consist of four sub directories, see the table below.

Directory	Description
BLKSLD	Blocks and slides belonging to the symbol library. Contains sub directories with blocks, slides and set-ups.
FORMAT	Definitions for the forms. Contains sub directories with blocks and set-ups.
PLOT	Definitions for the plotters. Contains sub directories with blocks and set-ups.
TEMP	Directory used for the temporary files of the application. Used only in the personal

MLUSER library

The MLUSER library is used for storing your own made symbols and setups for forms and plotters. Data in an MLUSER-tree is a “resource” that can be used for different purposes. You can use both local and shared MLUSER-trees (if you work in a network). How to best use the MLUSER-tree depends on if you work in a network environment, what type of drawings one wants to create and which applications you use.

Examples of how the MLUSER-tree can be used

Since the use of MLUSER-trees is highly dependant on how and what is being designed, one should see these examples as a one of many possible setups. Please do not hesitate to experiment to try and find a solution optimal to your own or your companies routines.

Example 1:
Single user, no network, only one MLUSER-tree.

1. Create a personal MLUSER-tree, for example C:\ALFA, with the sub directories BLKSLD, FORMAT, PLOT, CFG and TEMP.
2. Give the path to the MLUSER-tree with the MLUSER-variable, such as: ED_MLUSER1=C:\ALFA

This is the simplest case, recommended for single users working alone or if one is uncertain how to use the MLUSER-trees.

Example 2:
For users without networking capabilities who want to work with two different symbol libraries and/or plotter definitions. Except these two MLUSER-trees there is also a personal MLUSER-tree.

1. Create a personal MLUSER-tree, for example C:\ALFA, with the sub directories BLKSLD, FORMAT, PLOT, CFG and TEMP.
2. Create another two MLUSER-trees for the different symbol libraries, forms and plotter-definitions, for example C:\BRAVO and C:\CHARLIE with the sub directories BLKSLD, FORMAT, PLOT, CFG and TEMP.

With different values of the MLUSER-variable you can now choose which MLUSER-tree you want to use, or if you want to use both.

- In order to use the MLUSER-tree C:\BRAVO (and the personal directory) write: ED_MLUSER1=C:\ALFA;C:\BRAVO.
- In order to use the MLUSER-tree C:\CHARLIE (and the personal directory) write: ED_MLUSER1=C:\ALFA;C:\CHARLIE.
- In order to use both MLUSER-trees (and the personal directory) write: ED_MLUSER1=C:\ALFA;C:\BRAVO;C\CHARLIE.

Example 3:

Two users are working each with one computer on a network with a common server. They are both working with multiple MLUSER-trees. They each have a local hard disc (C:) and two common hard discs (J: and K:).

1. For each user, create a personal MLUSER-tree, for example C:\ALFA, with the sub directories BLKSLD, FORMAT, PLOT, CFG and TEMP. Create them on each local hard disc.
2. Create the wanted number of MLUSER-trees for the different symbol libraries, forms and plotter-definitions, for example J:\BRAVO, K:\CHARLIE and K:\DELTA with the sub directories BLKSLD, FORMAT, PLOT, CFG and TEMP.
3. The first user is working with the first and second MLUSER-tree. He or she enters the following in the MLUSER-variable:
   ED_MLUSER1=C:\ALFA;J:\BRAVO;K:\CHARLIE
4. The second user is working with the first and third MLUSER-tree. He or she enters the following in the MLUSER-variable:
   ED_MLUSER1=C:\ALFA;J:\BRAVO;K:\DELTA

These two users both have a personal MLUSER-tree (C:\ALFA), a common MLUSER-tree (J:\BRAVO) and a separate MLUSER-tree (K:\CHARLIE and K:\DELTA).

Command Reference

CHAPTER 8

In this chapter the commands that exist in elMaster design are described.

Most of the commands are available from the pull-down menu. In the cases where the commands also are available from the screen menu or the digitizer menu this is noted under the command.

elMaster Diagram and Layout have both common commands and specific commands for each part. To show which commands that are specific for Diagram respective Layout this is indicated with the following icons:

This icon shows that this command is specific for Diagram.

This icon shows that this command is specific for Layout.

The commands are listed in alphabetical order.

About *el*Master design

Shows text information about the *el*Master design application.

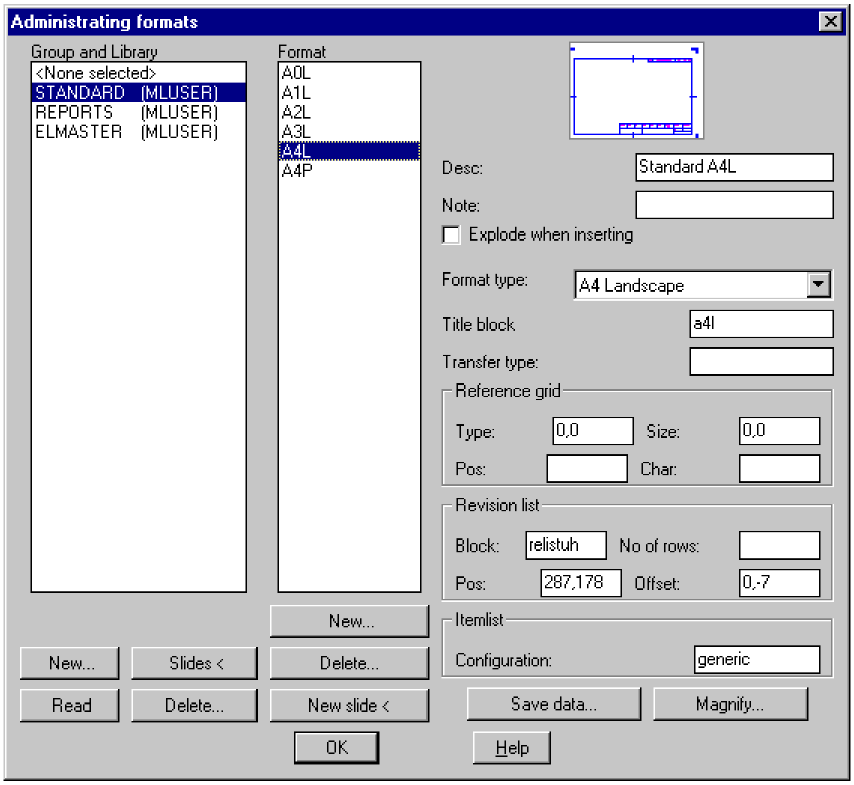
Admin (FORMAT)

This is the dialog box for the administration of the format and the format groups.

Screen menu: **FORMAT; Admin**

Work procedure:

1. Select the group from the group list. Data about the group can now be changed. You can now process the chosen group. Save the data about the group with the button **Save data**. The group data is copied as the initial value when the format is created in the respective group.
2. Select the format from the format list. Data about the format can now be changed. You can now process the chosen format. Save the data about the format with the button **Save data**.



Group and Library:
<None selected>
STANDARD (MLUSER)
REPORTS (MLUSER)
ELMASTER (MLUSER)

Format:
A0L
A1L
A2L
A3L
A4L
A4P

Desc: Standard A4L
Note:
☐ Explode when inserting
Format type: A4 Landscape
Title block: a4l
Transfer type:

Reference grid
Type: 0,0 Size: 0,0
Pos: Char:

Revision list
Block: relistuh No of rows:
Pos: 287,178 Offset: 0,-7

Itemlist
Configuration: generic

Figure 75. Administrating formats

Des:
Forms description.

Rem:
Possible remarks

Explode during insertion:
Here you can specify if the drawings title block shall explode during insertion.

Format type
Shows, which format type, belong to the form.

Reference net
Here you specify which type of reference net the formats have. You specify type of the net, size, position and what signs should be used.

Title block
Block name for title block

Revision list
Name and position for the list row. The number of rows specifies the maximum number of rows to be shown. Offset specifies the distance between the rows.

Item list configuration
Block name for item list configuration.

Save data...
Save your changes by pressing the button **Save data**.

Magnify...
Show the form’s icon (slide) magnified.

Group operations:

> **New**
> Create a new group.
>
> **Read**
> Reads the format block on the hard disk of a selected group. Use **Slides<** to create slides for the new block.
>
> **Slides<**
> Creates new slides for the format blocks in a group. You can choose if you want to create new slides for all the blocks in a group, or just for the blocks without slides.
>
> **Delete**
> Removes (deletes) an existing group. The group must not include any formats.

Block operations:

New
Create a new format block in a group.

Delete
Removes (deletes) a marked format block.

New slide<
Creates a new slide (icon) for a format block in a group.

More information:
See “Forms” on page 54.

Angle (DIMENSION)

Makes an angle dimensioning. This can be used in three different instances:

- Angle between two lines.
- Complete angle of an arc.
- Indicated angle of a circle.

Screen menu: **DIMENSION; Angle**

Digitizer menu: DIME**NSIONING; Angle**

Work procedure:

1. Select the first line (or at a point on an arc or circle).
2. Select the second line (if arc or circle has not been chosen).
3. Select the line's placement or, if an angle dimension has been made between two lines, press ENTER to estimate the large angle (180 degrees).
4. If dimensioning is done between two lines, the placement of the dimension line is determines which angle (of 4) to calculate.
5. Edit the dimension text.

More information:
See “Dimensioning” on page 91.

ANGLE (EDIT)

Draws a line in a given angle relative to a reference line.

Screen menu: **EDIT; ANGLE**

Digitizer menu: **EDIT; ANGLE**

Work procedure:

1. Select reference line.
2. Select start point of the line.
3. Select angle.
4. Select line length.

More information:
See “Editing” on page 94.

APERTURE

Sets the values of the variable APERTURE. The usage of APERTURE is described in the AutoCAD-manual.

Screen menu: **MLKONFIG; APERTURE**

Work procedure:

1. Select a new value for the variable APERTURE.

Arrow

Simplifies the drawing of arrows that point to the conductors specified by the symbol. It is used together with a symbol and is moved from the symbol to the conductors, which the symbol specifies.

Digitizer menu: **CABLE/CORD SPEC.**

Work procedure:

1. Point at the start point.
2. Point at the conductor, which the arrow shall point at, horizontally or vertically from the start point.
3. End the command with ENTER.

Attribute Angle

Changes the angle of the attribute.

Digitizer menu: **EDIT DESIGNATIONS.**

Work procedure:

1. Select a designation.
2. Specify a new angle.

Attribute Angle 0-Posit

Changes the angle of the attribute to zero and requests new insertion point.

Digitizer menu: **EDIT DESIGNATIONS.**

Work procedure:

1. Select a designation.
2. Point at the designation’s new insertion point.

Attribute Layer

Change the layer on an attribute. For example, if you want to remove a remark so that you can change the layer to the designers layer.

Digitizer menu: **EDIT DESIGNATIONS.**

Work procedure:

1. Select a designation.
2. In the dialog box, select the new layer.

Attribute Lgroup

Changes layer group on attribute, for example, layer D11 (constr. aid) to H11 (hole punch).

Attribute Position

Moves attribute.

Digitizer menu: **EDIT DESIGNATIONS.**

Work procedure:

1. Select a designation in a symbol.
2. Point at the designation’s new insertion point.

Attribute Text height

Changes the text height of the attribute.

Attribute Value

Edit value of attributes in blocks.

Base line (DIMENSION)

Draws a base line dimensioning originating from a vertical, horizontal, rotated or parallel dimension.

Screen menu: **DIMENSION; Base line**

Digitizer menu: **DIMENSIONING; Base line**

Work procedure:

1. If the last drawn entity is one of the above mentioned dimensioning types, specify the next extension point, otherwise select a new base dimensioning by pressing ENTER and pointing at it. The last drawn entity will be highlighted.
2. Edit the dimension text.

More information:
See "Dimensioning" on page 91.

Blocklib

More information:
See "Symbols" on page 63.

Admin

This function is used to manage the symbol directories where the block is placed. New directories can be created and existing directories can be removed.

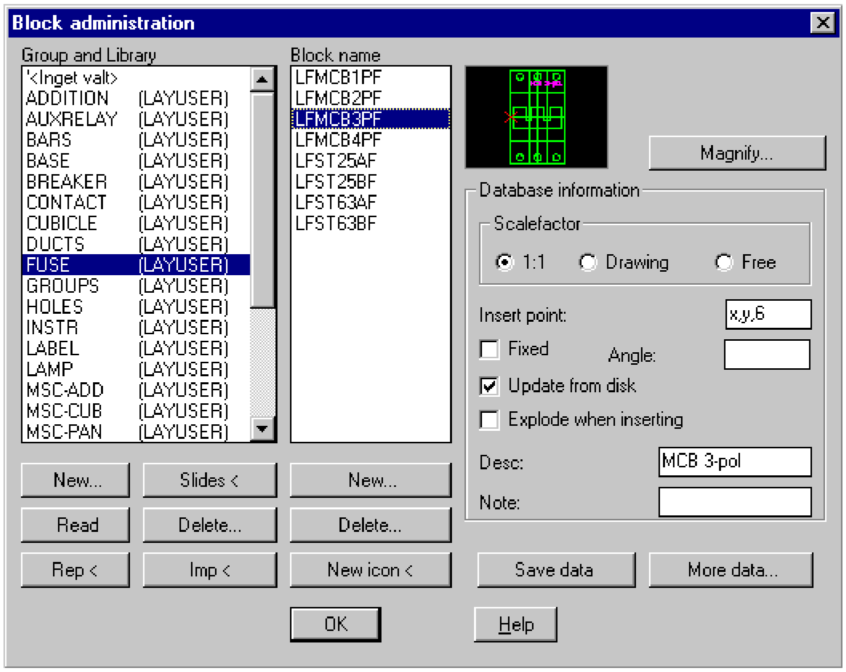
Data can be partly connected to a group and partly to individual blocks. Every group has certain insertion properties (scale factor, angle etc.), which apply to all the blocks in a group. If you specify properties for an individual block, these properties will override the original properties of the group.

Screen menu: **BLOCKLIB; Admin**

Digitizer menu: **BLOCKLIB; Administrate directory**

Work procedure:

1. Mark the directory that you want to change.
2. Change the data in the field and save the settings by pressing Save data.
3. Finish by pressing **OK**.



Block administration

Group and Library	
'<Inget valt>	
ADDITION	(LAYUSER)
AUXRELAY	(LAYUSER)
BARS	(LAYUSER)
BASE	(LAYUSER)
BREAKER	(LAYUSER)
CONTACT	(LAYUSER)
CUBICLE	(LAYUSER)
DUCTS	(LAYUSER)
FUSE	(LAYUSER)
GROUPS	(LAYUSER)
HOLES	(LAYUSER)
INSTR	(LAYUSER)
LABEL	(LAYUSER)
LAMP	(LAYUSER)
MSC-ADD	(LAYUSER)
MSC-CUB	(LAYUSER)
MSC-PAN	(LAYUSER)

Block name
LFMCB1PF
LFMCB2PF
LFMCB3PF
LFMCB4PF
LFST25AF
LFST25BF
LFST63AF
LFST63BF

Database information

Scalefactor: ● 1:1 ○ Drawing ○ Free

Insert point: x,y,6

☐ Fixed Angle:

☑ Update from disk

☐ Explode when inserting

Desc: MCB 3-pol

Note:

Figure 76. Block administration

Scale factor

You can choose if you want to insert a block with the scale of 1:1 in the drawing’s scale factor, or with an "free" scale factor (the program will prompt you during every insertion).

Insertion point

Where you specify if the x-, y- or z-co-ordinates are to be pre-selected.

Example: “x,y,z” will make the program ask for all three co-ordinates. “x,y,0.0” will make the program ask for the x- and y- co-ordinates whilst the z- co-ordinate is always specified at “0.0”. If you specify “0,0,0”, the symbols are set without you being asked about the insertion point.

Angle

You can choose if you want to insert a block with a pre-determined insertion angle (**Fixed**) or if you want the program to ask for the insertion point before every insertion.

Update from disc

You can choose if you want to update the block from the hard disc (with **insert**) or to make a copy from the drawing if the block is already inserted in the drawing.

Explode during insertion

You can choose to insert the block "exploded" i.e. inserted in loose parts with (with **insert**). In this case there will be no block definition created.

Block names

Here you can save the name of a block.

Remarks

Here you can save complementary text about the block.

Category

Here you can save to which category the block belongs. For example symbols (SYM) or object (OBJ).

Group operations:

> **New...**
>
> Creates a new group.
>
> **Delete...**
>
> Removes an existing group. The group must not include any blocks.
>
> **Slides<**
>
> Creates new slides for a block in a group. You can choose if you want to create new slides for all the blocks in a group, or just for blocks without slides.

Scan

Registers the blocks that are on the hard disc of a selected group. Use **New slides** to create slides for a new block.

Rep.<

Shows a dialog box where you can create a report (chart) with a block from a group. Se “Setting reports (Rep.<)”, page 149, for more information about the dialog box.

Imp.<

Shows a dialog box where you can imports block and/or block information from a report (chart) to an existing group. See “Import setting (Imp.<)”, page 150, for more information about the dialog box.

Block operations:

New...

Create a new block in a group.

Delete...

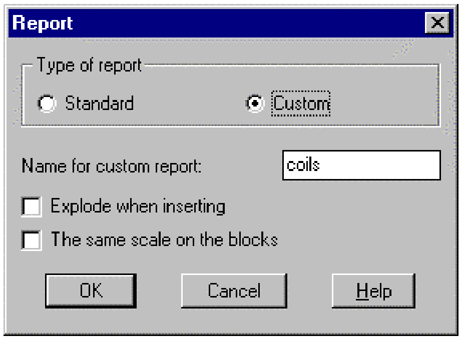
Deletes a marked block.

New icon<

Creates a new slide for a block in a group.

Setting reports (Rep.<)

The function creates a frame around every block. The frame includes all the information about the block including the block name. In the frame there is also a block which marks the block's insertion point.


Report

Type of report

○ Standard ● Custom

Name for custom report: coils

☐ Explode when inserting

☐ The same scale on the blocks

Figure 77. Report

Standard

Creates a standard report with a frame around every block in the group. The data in the database record is stored in a hidden attribute in the block. All the blocks in a group are inserted in the same drawing.

Custom

Creates a customized report. The function inserts the block in a pre-determined form.

Name for custom report

Only used together with **Custom report**. Specifies the prefix in the file name for the drawings, which should be saved. For example: “symbol” is saved as SYMBOL.DWG, SYMBOL2.DWG... SYMBOLN.DWG. Up to six characters can be specified in the file name prefix.

Explode when inserting

Specifies if the block shall be exploded when inserting in the report.

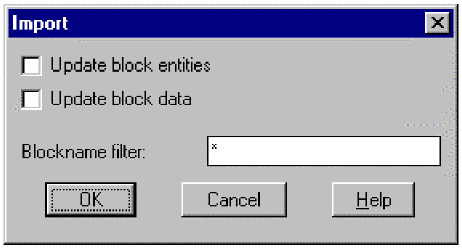
The same scale on the blocks

Only together with **Custom report**. When this is marked the largest symbol decides which scale the symbols should be inserted with. In other case each symbol is inserted with the largest possible scale.

Import setting (Imp.<)

You can choose if the block and/or block information shall be imported. A block, which is not already in a group, is always imported with both the block and the information.

Use **New slide** to create slides for the new block.



☐ Update block entities
☐ Update block data
Blockname filter: *

Figure 78. Import

Update block entities

Update the block’s graphical representation (entities). Update data about a block. Update database record with data from the report.

Block name filter

Specifies a filter, which is used to choose a block from a report. Only the blocks, which name matches the filter, will be imported.

Note: The blocks in the report, which are not already in the group, will always be imported with both graphics and data (if they have a block name which matches the filter).

New frame

Creates a new frame, which is used to add a new block to a report created with BLOCKLIB.

Screen menu: **BLOCKLIB; New Frame**

Work procedure:

1. Specify **Lower left corner** and **Upper right corner** in the frame.
2. Specify the block name (this must be filled in).

New Insert point

This function creates a new marking of the block’s insertion point. You can only have one insertion point in a frame. Other points are ignored.

Screen menu: **BLOCKLIB; New Insp.**

Work procedure:

1. Specify the insertion point for the block.

Brackets

Creates a bracket to be used, for example, around a number of conductors with references. This bracket is only “draft”.

Digitizer menu: **REFERENCES**

Work procedure:

1. Point to the top/left end of the bracket.
2. Point to the bottom/right end of the bracket.
3. Specify the direction of the bracket relative to one end (start or end point).

Break Connection

This function adds a corner point on a connection line, mechanical link or a boundary frame.

Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Enter point/corner on connection/boundary frame.
2. Enter new insertion point.

Break line (DIMENSION)

Breaks up extension lines when these cross each other or a dimension line.

Screen menu: **DIMENSION; BREAK LINE**

Digitizer menu: **DIMENSIONING; Break line**

Work procedure:

1. Select the extension that should be broken.
2. Specify the point at which the line should be broken, or press ENTER to break the extension line towards a text.

Note: You cannot break dimension lines, but you can break extension lines that are crossed by a dimension line. If you want to break a dimension line, you must first explode the dimension. For example, use the function LEXPLODE.

More information:

See “Dimensioning” on page 91.

Cable

Draws cable symbol in a connection diagram. Note that this symbol is not linked to *el*Master Cable; it is only a “draft” symboly.

Begin by pointing to the start point, which is the start point of the first conductor. Specify the start point of the last conductor as the second point. The distance between these points is a multiple of 5 (5 = distance between box terminals). This is easy since grid is 5.

In this command you do not specify the number but the extent, and this automatically gives the number. The direction and length of the line to which all conductors are to be brought together is then specified as a line of direction and length orthogonal with respect to the start point of the first conductor, mentioned above. Finally, specify the start number for the first conductor; the conductors are then numbered automatically.

Digitizer menu: **CONN.LINES**

Work procedure:

1. Point at the first start point.
2. Point at the last end point.
3. Specify direction and length.
4. Specify start number and numbering.

CENTERBOX (DRAWBOX)

Draws a rectangle according to the specified center point and corner. The function marks a quadrant of a rectangle, which in turn enables the entire rectangle to be drawn.

Screen menu: **DRAWBOX; CENBOX**

Digitizer menu: **DRAWBOX; Box 2**

Work procedure:

1. Enter the center point of the rectangle.
2. Enter one corner of the rectangle
3. Enter the angle.

CHAMFER (EDIT)

Draws a chamfer with the help of the AutoCAD command **Chamfer**. Rather than giving the distance of both sides, this function requires only that the distance on one side of the chamfer angle be given.

Screen menu: **EDIT; CHAMFER**

Digitizer menu: **EDIT; CHAMFER**

Work procedure:

1. Point at the first line to be chamfered.
2. Point at the second line to be chamfered.
3. Select chamfer width.
4. Select chamfer angle.

More information:

See “Editing” on page 94.

Change all symbols

This function changes all the inserted symbols in a drawing that has the same symbol name in the symbol directory, which has been set in the configuration. This function can be used, for example, when you want to go from one symbol standard to another.

Change scale factor (FORMAT)

Changes the drawing's scale factor. The function also changes the scale factor of all variables that are dependent on the scale factor and, if desired, can re-scale the drawing frame.

Screen menu: **FORMAT; SCALE; Change**

Note: Only the drawing frame will be re-scaled. The other objects of the drawing will not be re-scaled. Refer to the AutoCAD command SCALE for more information about how to re-scale the other objects of the drawing.

Change selected symbols

This function can be used when you want to change one or several symbols to any other symbol, with another appearance, but the same values on the designations. The function can, for example be used when you want to change a make contact to a break contact. The dialog box (and how you use it) is the same as for the one used when inserting a new symbol. The first symbol in the selection will determine which symbol group that will be shown in the dialog box (assuming that the selected symbol exists in any symbol group).

Work procedure:

1. Select the block you want to change.
2. Select the new block in the dialog box.

Change style

Changes the “text font” (appearance of letters) of “draft” text. The function changes text which is created with the styles STANDARD, TERMINALS, ITEMDES.

- Single gives AutoCAD’s standard font Txt.
- ISO gives AutoCAD’s standard font Iso.
- Simplex gives AutoCAD’s standard font Simplex.

Digitizer menu: **DECORATIVE TEXT.**

Change texts

Is used when part of a text is to be revised. This is easily done by selecting the string of text that you want to revise and then specifying its new value.

Digitizer menu: **DECORATIVE TEXT**

Work procedure:

1. Point at the text that you want to revise.
2. Specify the string of text that you want to revise.
3. Replace with the revised/new text.

Change Value global

This function is similar to the function ATTEDIT in AutoCAD and it can be used when you want to change the attributes of several symbols. The reason that you can not use the ATTEDIT function is that the attributes are stored in a special way. Values are stored as extended data (x-data), while the attributes only are used to display the values. If you change the value in the attribute directly, for example with AutoCAD's ATTEDIT, the x-data value will not be changed.

The selection can be filtered, as with ATTEDIT, on block name, attribute type (i.e. TAG in ATTEDIT) and value. By selecting "*" in block name, attribute type, or value, everything will be selected.

- Block name <*>:
- Attribute type <Function/Placement/Product/Terminal/Signal <*>:
- Value <*>:
- Text that you want to change:
- New text:

CIRCLE (EDIT)

This command breaks up an arc or circle. The arc segment between a given start angle and end angle are taken away. The break occurs counter clockwise.

Screen menu: **EDIT; CIRCLE**

Digitizer menu: **EDIT; CIRCELBREAK**

Work procedure:

1. Select the object to break.
2. Select the start angle.
3. Select the end angle.

Clear (FORMAT)

All objects on the layer, which is specified in the drawing's configuration (format layer), are cleared and the drawing's format settings are restored.

Screen menu: **FORMAT; Clear**

Note: The data in the drawing's title block will be lost.

CMDDIA (MLPLOT)

This variable specifies whether or not to work with a dialog box while plotting with AutoCAD's ordinary Plot command (1=dialogbox, 0=command from the keyboard).

Screen menu: **MLPLOT; CMDDIA**

Digitizer menu: **FORMAT; MLPLOT; CMDDIA**

More information:
See "Plotting" on page 98.

Common itemdes.

When all the information on a sheet is in the same unit, this “information block” is used to give all other units a prefix, which, for example, could be a container name. The function searches the drawing for earlier inserted item designations. If none is found, a new one is inserted.

Digitizer menu: **BOUNDARY FRAMES**

Work procedure:

1. Place out the block.
2. Specify the value for the common designation.

Cond.group

This command is used on connection lines to specify the number of conductors. It is most often used in single-line diagrams. It can be used in two ways:

- Number of “slashes” indicates the number of conductors.
- One “slash” and a number indicate the number of conductors.

Digitizer menu: **CABLE/CORD SPEC.**

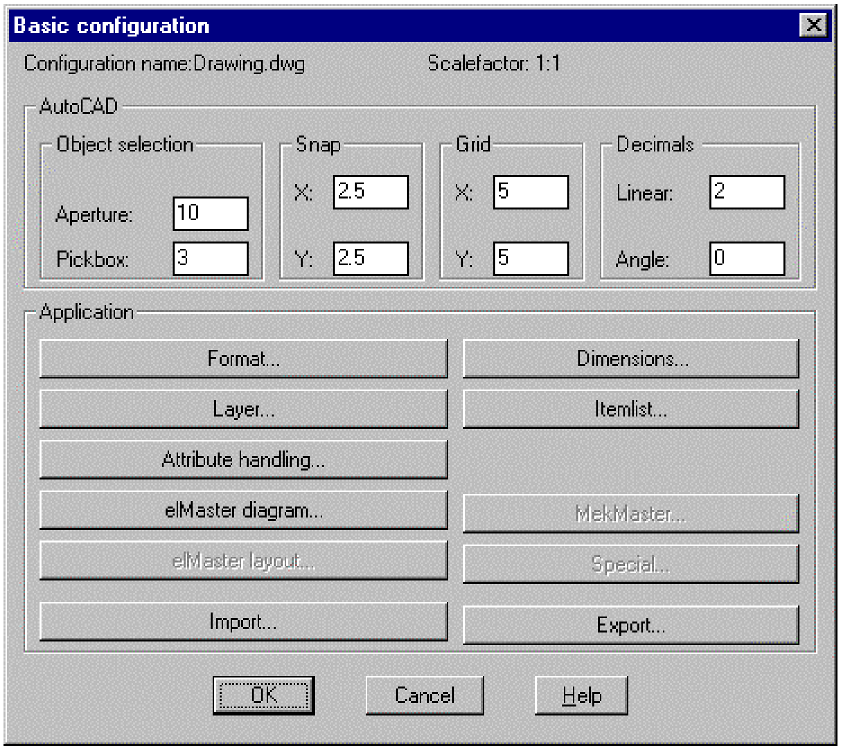
Work procedure:

1. Point to a point on the line.
2. Specify the angle of the conductor (0 or 90 degrees), default is 0 degrees.
3. Specify the number of slashes.
4. Specify a number if the number of slashes was one.

Config.

The dialog box looks different (some texts are not lighted) depending if you open it in *el*Master Diagram or Layout.

Screen menu: **MLCONFIG; CONFIG**



Configuration name:Drawing.dwg
Scalefactor: 1:1

Aperture: 10
Pickbox: 3
X: 2.5
Y: 2.5
X: 5
Y: 5
Linear: 2
Angle: 0

Figure 79. Basic configuration

Configuration name

Name of the drawing when the application was started. Used to determine which configuration is the real one if the drawing by mistake gets many configurations.

Scale factor

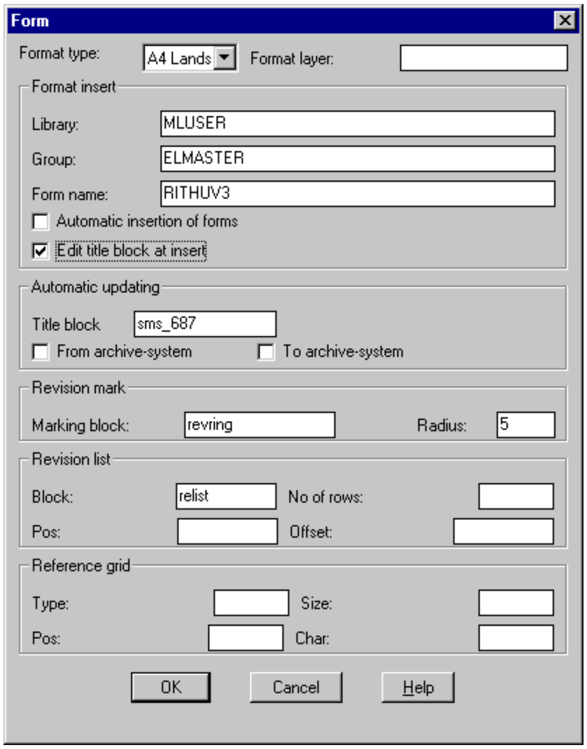
The drawing’s current scale factor.

Format

A format has several settings linked to it. Some of these are stored in the database records, which are connected to the format. These settings are transferred to the drawing's configuration when a form is inserted.

Here you find the information about the format type in the inserted drawing form. The format types are registered by the PLOT function among others. You can also govern how the form is drawn, automatically updated and how the revision markings will look.

Screen menu: **FORMAT; Config.**



Form

Format type: A4 Lands

Format layer:

Format insert

Library: MLUSER

Group: ELMASTER

Form name: RITHUV3

☐ Automatic insertion of forms

☑ Edit title block at insert

Automatic updating

Title block: sms_687

☐ From archive-system ☐ To archive-system

Revision mark

Marking block: revring

Radius: 5

Revision list

Block: relist

No of rows:

Pos:

Offset:

Reference grid

Type:

Size:

Pos:

Char:

Figure 80. Form

Format type

Information about the format described in the standard that is used.

Format layer
Layer, in which the form is inserted.

Directory, Group and Form name
A predefined choice of form, which is used with automatic insertion of form.

Automatic insertion of form
Specifies if a form shall be inserted automatically during the opening of a drawing.

Edit form data when inserting
Specifies if the dialog box for editing data shall be shown when you insert a form.

Title block
Name of the block that includes data in the drawing's title block. This block name is used to automatically find (read/write) data in the drawing's title block.

From the archive system
Specify automatic updating of the drawing's title block.

To the archive system
Specify automatic updating of the archive system.

Marking block
Block name for revision marking.

Radius
Radius (size) for revision marking.

Revision list, block
Block name for revisions list row.

Revision list, position
Insertion point (x, y) for first revision list row.

Revision list, number of rows
Maximum number of revision rows, which should be shown in the drawing.

Revision list, offset
Distance between revision list rows.

Reference net, type
Specifies the number of squares in x- and y-direction.

Reference net, size
Specifies the size on respective square in the reference net.

Reference net, position
Specifies the distance between the reference net's origin of coordinates to the drawing's origin of coordinates.

Reference net, characters
Specifies the characters that are going to be used to describe the reference net. It also specifies the direction of the reference net. A minus sign before the character turn the direction, which normally is left to right and from top to bottom.

Layer

For marking the layers, which do not belong to the active layer group and thus locking them. Non-active layers are locked or unlocked automatically if this setting is changed.

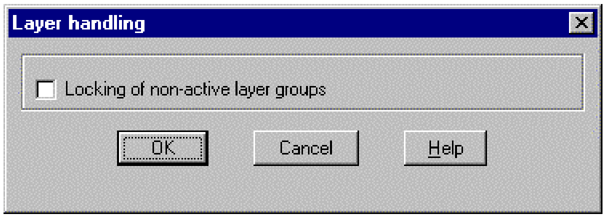


Figure 81. Layer handling

Attribute handling

This dialog box includes functions to guide editing attributes together with a block in BLOCKLIB.

The settings make it possible to receive questions about attributes and if they should appear in the dialog box or the command row.

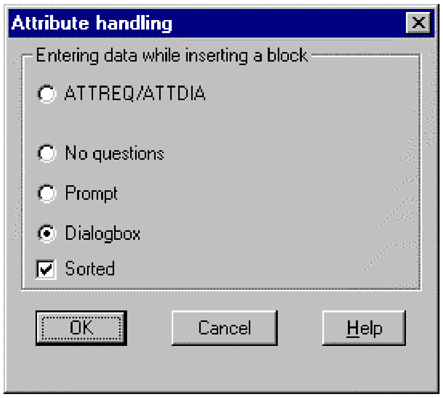


Figure 82. Attribute handling

ATTREQ/ATTDIA

Uses AutoCAD’s variables ATTREQ/ATTDIA and AutoCAD’s dialog box (DDATTE).

No questions

No questions about the attribute are asked. ATTREQ/ATTDIA ignores.

Prompt

Uses BLOCKLIB’s function to ask about an attribute. ATTREQ/ATTDIA ignores. The question is asked via the command row.

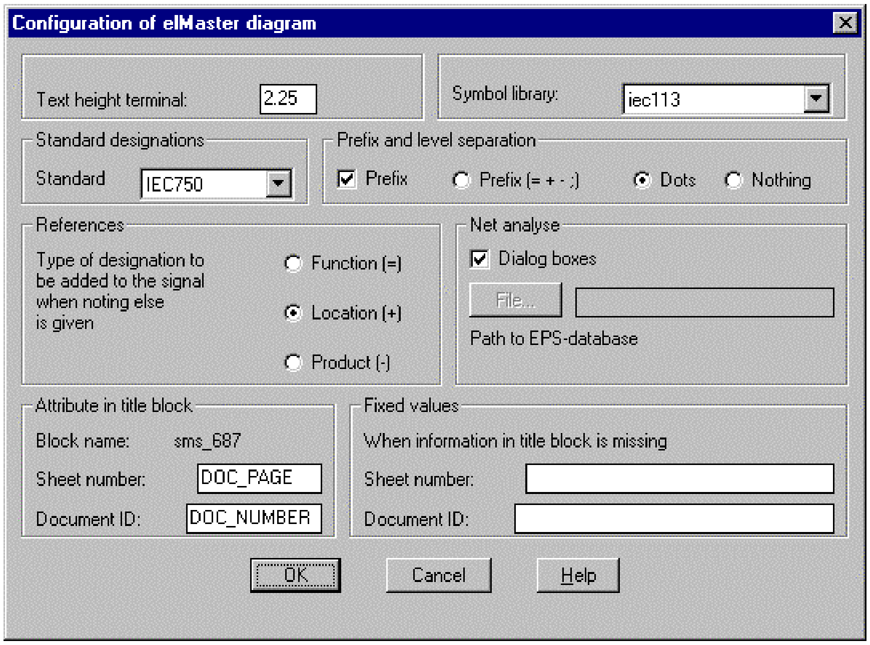
Dialog

Uses BLOCKLIB’s function to ask about an attribute. ATTREQ/ATTDIA ignores. The question is asked via the dialog box.

Sorted

Alternatives **Prompt** and **Dialog** have further possibilities to help. The settings outside the dialog box make it possible to guide the questions, including the order in which they are asked etc.

*el*Master Diagram



Configuration of elMaster diagram

Text height terminal: 2.25

Symbol library: iec113

Standard designations

Standard IEC750

Prefix and level separation

☑ Prefix ○ Prefix (= + - :) ● Dots ○ Nothing

References

Type of designation to be added to the signal when noting else is given

○ Function (=)

● Location (+)

○ Product (-)

Net analyse

☑ Dialog boxes

Path to EPS-database

Attribute in title block

Block name: sms_687

Sheet number: DOC_PAGE

Document ID: DOC_NUMBER

Fixed values

When information in title block is missing

Sheet number:

Document ID:

Figure 83. Configuration of elMaster Diagram

Text height terminal

The selected text height only affects the symbols when they are inserted. To adjust the text height of symbols which are already inserted. See “Text height terminals” page 258.

Symbol library

During the setting procedure, the **symbol directory** function selects which of the two symbol directories will be applicable for the selection of the symbol from the pull-down menu or digitizer menu.

Standard designations

The standard for item designations governs, amongst other things, which prefix is permitted. This in turn steers the net analysis uniting of complete item designations.

Prefix and level separation

Prefix and separation characters affect the way that item designations are shown in a drawing.

References

Settings for references are governed by which type of item designation the reference will be associated with, if nothing else is specified.

Net analyse

When setting the net analysis, it is possible to switch of the user interface for the EPS-function. If this is done, then the search path and name of the EPS-file must be entered.

Attribute in title block

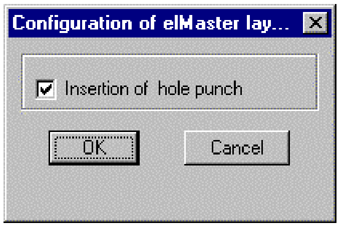
Attributes in the drawing’s title block alternatively fixed values for document identification and sheet number.

Fixed values

Attributes for document identity and sheet number in the title block of a drawing must be entered before the EPS-function can find this information. The EPS-function uses both these values so that the EPS-database can create unique items. If there is no title block, or if the title block does not include a field with this information, then it is possible for you to specify a fixed value for the document identity and the sheet number.

*el*Master Layout

Opens a dialog box where you can check if the hole punch should be inserted together with the symbol.



☑ Insertion of hole punch

Figure 84. Configuration of elMaster layout

If **Insertion of hole pattern** is on, the hole-pattern that is defined in the attribute symbol PUNCH will be inserted. The hole-pattern should be a symbol in the symbol library HOLES.

Module Units

In the functions **Free symbol**, **Copy line** and **Length bars** it is possible to specify certain measurements in **Number of module units**. In the file LAYINIT.LSP, the module system to be used is defined as well as the corresponding millimeter measurement. To add, change, or make corrections in this file, the basic adjustment is:

S-module = 44.45 mm

C-module = 7.0 mm

Dimensions

Opens a dialog box for setting of procedure for dimension setting.

Distance between object and dimension line

You receive the pre-set value for the distance between the object and the dimension line if you press ENTER when prompted about the placing of the dimension line.

Distance between the dimension line (DIMDLI)

Distance between two dimension lines. See AutoCAD-variable DIMDLI.

Simplified base line dimension

Marks if the text shall be parallel or angled towards the dimension line when you create the simplified base line dimension.

Item list

This is where you can configure the item references and the appearance of the item list.

Block name

Block name for the item references.

Radius

Radius for the item references.

Item references shape

Specifies the shape of the block used for item references.

Type of connection

Auto, **Arrow**, **Dot** or **Dash**. If the alternative **Auto** is selected, the function automatically changes between **Arrow** and **Dot** depending on if you choose a point close to an object or an isolated point.

Arrow block

Block name for arrow point.

Title row block name

Block name for the block, which is used as the item references title.

Row block name

Block name for the block, which is used as the item reference row.

Template

Name of the file, which should be used as template. See the AutoCAD-manual for a more detailed description of the template file.

Dist. tit. row:

Height of the block, which is used as item reference.

Dist. row:

Height of the block which is used as the item reference row.

Sum columns

Specifies if the positions, which have the same position number, should be shown in one or more rows in the item references.

Columns to sum

Specify which column the number is found in. This setting depends on which template is used.

Export

Prints some of the configuration information available in the drawing to a file with the extension CFG. This file can then be registered with the import function.

Import

Registers a configuration, printed out with the **Export** function, and updates the configuration of the drawing with this information.

CONNECT (EDIT)

Connects a line to another with a given radius. Wherever applicable, the command handles extending and shortening of the connection line.

Screen menu: **EDIT; CONNECTION**

Digitizer menu: **EDIT; CONNECTION**

Work procedure:

1. Select the line to be connected. Point close to the end point to be connected.
2. Point out the line for which the new line should be connected to.
3. Select the radius of the arc.

CONTINUE (DIMENSION)

This function draws a CONTINUE-dimensioning setting originating from a vertical, horizontal, rotated or parallel dimension.

Screen menu: **DIMENSION; CONTINUE**

Digitizer menu: **DIMENSIONING; CONTINUE**

Work procedure:

1. If the last drawn entity is of one of the above mentioned dimensioning types, specify the next dimension extension point or enter a new base dimension by pressing ENTER and point at it. The last entity will be shown by highlighting.
2. Edit the dimension text.

More information:

See “Dimensioning” on page 91.

Connect symbols

In order for cross-references, between apparatus different parts, to be generated, the apparatus must have a main symbol, which must be connected with its cross-reference symbols. This function creates these connections.

The main symbol is the symbol that corresponds to the operating device of the apparatus. In a relay, this corresponds to the symbol for the coil. The main symbol must have the symbol type set to ED_MAIN. There can be only one main symbol in an apparatus. There can be more than one cross-reference symbol, but these must be in the same drawing as the main symbol. A cross-reference symbol must have the symbol type set to ED_XBLK.

When you are going to select the symbols that you want to connect, you can do this either by selecting them from the drawing by pointing at them, or by selecting a window that includes the symbols. If the selection contains more than one main symbol, no main symbols or no cross-references, there will be an error message and no connection will be created.

Conn.Pt

Places a connection point with a cross in a boundary frame serving as a marobox/function unit or “black box”. The frame must be given a “draft” content (“draft” symbol or text). This command can be used to create connection points to the macro symbol. The same effect can be obtained with terminals but this symbol is more convenient since it is a ready-made unit.

Digitizer menu: **BOUNDARY FRAMES**

Work procedure:

1. Point to insertion point within the box.
2. Specify the angle.
3. Specify the location designation if the designation is not yet complete.
4. Specify the terminal designation.

Contact style.

Changes the appearance/style of contacts, with or without “ring”.

Screen menu: **MSC.SET; Contact style**

Digitizer menu: **SETTINGS**

Convert

_el_Master 12=>14

This function changes the drawing so that it corresponds to the number of layers, attributes and so on, for the current version of the product. This function also handles drawings created in versions previous to 12.

_el_Master LT=>14

This function handles drawings made in LT *el*Master 3. The difference between a drawing in LT *el*Master and *el*Master design is that a drawing in LT *el*Master lacks information about which structure a designation belongs to. This information is important when the drawings are being used as a base for apparatus lists and connection tables.

The function uses the prefixes that are supposed to be used according to the IEC 750 standard when deciding which type of designation a value is associated with. According to the IEC 750 these prefixes are "+" for location, "=" for function, and "-" for product. If a designation lacks a prefix, this function will assume that the value in field ID1 is the location designation, the value in ID2 is the function designation and the value in ID3 is the product designation.

Convert Blocklib

Converts a symbol created in an earlier version of *el*Master.

Create a standard report with symbols that you want to convert. Make sure the symbols are not "exploded". Start the function. When the conversion is complete, the symbols can be "read" with the help of the Blocklib function **Import**.

Copy One

Copies a symbol and lists the item designation.

This function also inserts a hole punch if the configuration **Insertion of hole punch** is switched on and the hole punch reference has been marked.

The stepping-up of the item designation proceeds according to the following rules:

If the item designation is a number, the new value is equal to the item designation of the original together with the value of the increment item designation. For example, "1" to "2", "100" to "101" and so on.

If the item designation consists of a letter and a number, the value of the increment is added to the numeric part. For example, "K1" to "K2", "RS100" to "RS101", and so on.

If the original item designation ends with a letter there will be no stepping-up for the copy.

Copy Row

Creates a matrix of the selected object and steps-up the item designation.

The function also inserts hole punch if the **Insertion of hole punch** setting is on and the hole punch reference has been marked.

To create a row of copies, the function must know two things: the number of copies in x- and y-length, and the distance between the rows. The values for this information can be entered in various ways into the dialog box.

Alternative 1

Specify an area to be filled with copies and indicate the desired distance between copies. The number of copies that fit into the area is automatically calculated.

Alternative 2

Specify the number and the distance between the copies.

The space to be filled and the distance between can be given in three different ways:

- By pointing out two points in the drawing.
- By pointing at a visible line in the drawing. For example, if you point at a terminal, the copies will be placed without any space between them (i.e. the C-C measure will be the same as the width and height of the terminal.).
- By specifying a measurement in millimeters, inches or a number of module units.

Stepping-up of the item designation is done in the same way as for the **Copy One** function described above.

If the increment for the item designation has been marked earlier, its value is shown in the text line at the bottom edge of the dialog box.

CROSS HEIGHT (BLOCKLIB)

Sets the size of the cross that marks the insertion point when block icons (slides) are created. The size 0.0 means that a cross will not been drawn out.

Screen menu: **BLOCKLIB; Cross height**

Work procedure:

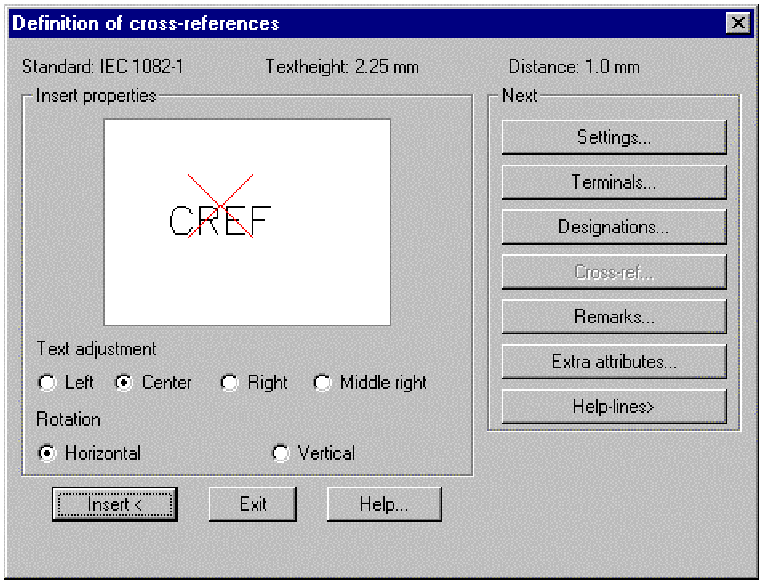
1. Select the height the cross should have in relation to the icon the (slide).

More information:

See “Symbols” on page 63.

Crossref.

With help of this dialog box you can create and insert a cross-reference attribute (CREF) when creating a new symbol.



Definition of cross-references

Standard: IEC 1082-1

Textheight: 2.25 mm

Distance: 1.0 mm

Insert properties

Text adjustment

○ Left ● Center ○ Right ○ Middle right

Rotation

● Horizontal ○ Vertical

Figure 85. Definition of cross-references

Insertion properties
Select the required text type and rotation.

Insert
Insert all cross-reference attributes.

Database error list

Shows a list that contains all possible errors that may appear in the functions **Save to database** and **Update from database**.

DECIMALS (MLKONFIG)

Changes the number of decimals that AutoCAD shows and works with in texts. The setting is stored in the AutoCAD variable LUPREC.

Screen menu: **MLKONFIG; DECIMALR**

Work procedure:

1. Select the number of required decimals.

Note: The AutoCAD variable DIMZIN influences how the decimals are copied. With different setting in DIMZIN, it is possible to suppress zeros before and after the decimal point. For further information refer to the AutoCAD manual.

This setting does not affect the accuracy of AutoCAD object handling. This is always done with precision.

Design data (interconnect)

Design data is information in the circuit drawing about how the drawing should be designed. The information is used by *el*Master interconnect.

Conn. control

Connection control is a function for describing how an electrical point in a circuit diagram should be connected. The terminals are numbered in the order that they should be connected. A terminal is connected with all the terminals in an electrical point, which is numbered with a number that is one higher or one lower then its own number.

Radial

Radial connection numbers the first terminal with a number and the rest with one number higher. All terminals are connected with the first.

Serial

Serial connection numbers the terminals in numerical order with the start on the given number.

Wire def. and Part def.

Part def. connects an article or an object in the database to a specific connection.

Kabel def. connects an article or an abject in the database to a specific cable.

If no value is given after a connection is specified, information about the connection in the drawing is shown after updating from database. It is not possible to change connections that are saved in the database from *el*Master design.

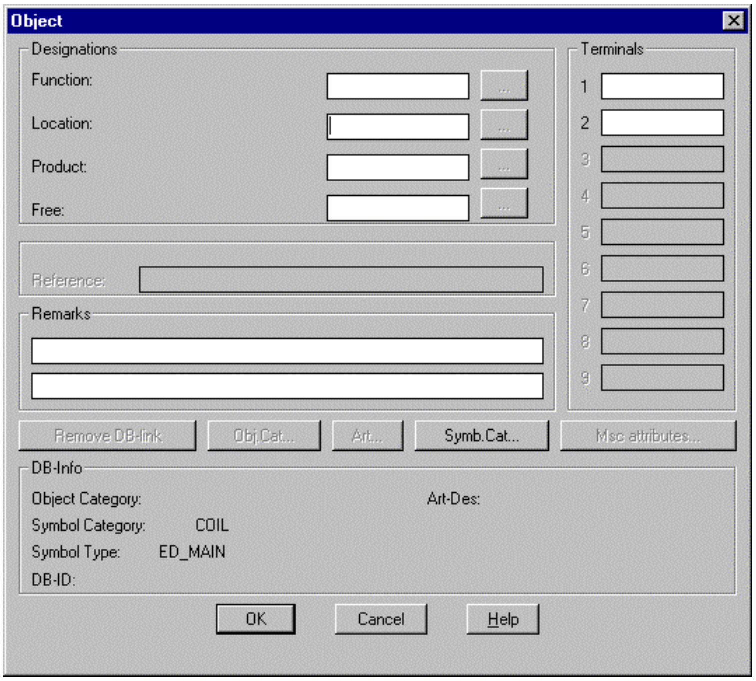
Work procedure:

1. Select start point (point where the information about the Cable/Wire will be displayed).
2. Specify cables/wires by selecting the lines representing the cables or wires. If you select a point in the drawing with no object, all lines that

intersect the line between the two last selected points will be selected at the intersection points.

3. End with ENTER.

The dialog box shown below will appear:



Symbol Category: COIL

Symbol Type: ED_MAIN

Figure 86. Object

Art

With this function you can assign a connection a new cable or wire.

The information about the cable or wire will not be displayed until the drawing is both **saved to database** and **updated from database**.

...

With any of these buttons you can make an object lookup to assign a connection an existing cable or wire.

The information about the cable or wire will not be displayed until the drawing is both **saved to database** and **updated from database**.

Design data connection (Connect)

Connection point data is information that can be a link to a connection point in a symbol.

The connection point data is inserted in the form of a block, which gives the value and describes the type of data and it’s value.

Work procedure:

1. Specify the insertion point on a connection.
2. Enter the value.

Design data net (Connect)

Net information is information that can be linked to a net (a point with common potential). Note that terminal and net are shared in *el*Master since terminals must have a 1-side and a 2-side.

Net information is inserted in the form of a block, which is given certain values that describe both types of data and their values.

Work procedure:

1. Point to the insertion point on a conductor.
2. Enter the values.

Design->Draft

This function checks if objects are in layers, which are “design”, and if you wish, moves them to the “draft” layer.

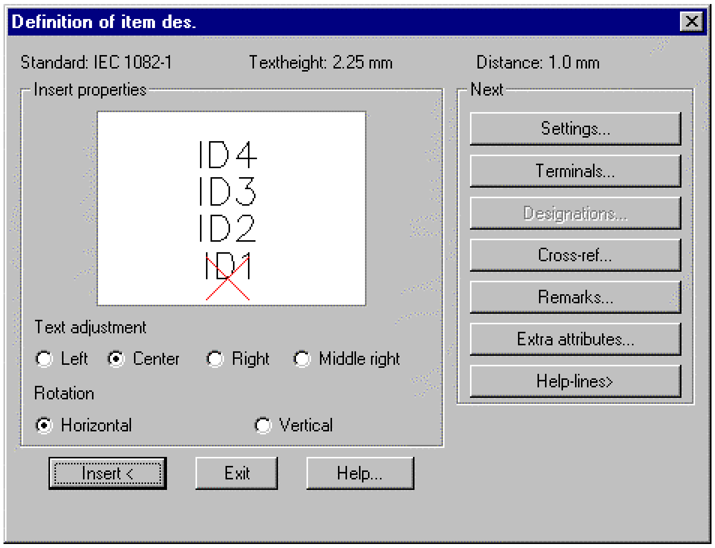
Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Select the objects to be checked/changed. The objects, which are connection-giving will be marked.
2. Cancel or select the objects to be changed. ENTER changes the objects selected in step 1.

Designations

With the help of this dialog box you can create and insert designation attributes (ID1-ID4) when you create a symbol. Which attribute you use to indicate a special designation, depends on how many and which designation is entered. The ID1 should always be positioned nearest the symbol.



Definition of item des.

Standard: IEC 1082-1 Textheight: 2.25 mm Distance: 1.0 mm

Insert properties

Text adjustment

○ Left ● Center ○ Right ○ Middle right

Rotation

● Horizontal ○ Vertical

Figure 87. Definition of item designations

Insertion properties
Select the required text type and rotation.

Insert
Insert all four attributes.

Designations on/off

Switches on/off designations for a symbol. The function switches off designations that are on and switches on designations that are off.

Switches on/off designations for a symbol. This is used, for example, when you add a terminal block in a row and will switch off the designation.

To switch off a single designation, for example, only the function designation, use the function **Attribute Layer**.

Note: The designations are not deleted, the function only changes the layer.

Digitizer menu: **EDIT DESIGNATIONS**

Dia1 (DIMENSION)

Makes a diameter dimensioning and places the dimension text in the center between the dimension arrows.

Screen menu: **DIMENSION; Dia1**

Digitizer menu: **DIMENSIONING; Dia1**

Work procedure:

1. Select circle or circle arc to be dimensioned. Dimensioning can be inserted in different angles depending on the selected object.

More information:
See “Dimensioning” on page 91.

Dia2 (DIMENSION)

Makes a dimensioning diameter with the dimension line drawn from the center and extended an arbitrary distance.

Screen menu: **DIMENSION; Dia2**

Digitizer menu: **DIMENSIONING; Dia2**

Work procedure:

1. Mark the arc or circle that should be dimensioned.
2. Edit the dimension text.
3. Point out the length of the leader.

Note: With the AutoCAD variable DIMTOH it is possible to decide whether the leader should extend a horizontal “tongue” for the dimension text to be printed on. Use the “Setvar” command to change this setting.

DIMTOH = 0 the “tongue” is not drawn.

DIMTOH = 1 the “tongue” is drawn.

More information:

See “Dimensioning” page 91.

DIMSTYLES (DIMENSION)

Commands for control dimensioning variables and DIMSTYLES.

Screen menu: **DIMENSION; DIMSTYLES**

Digitizer menu: **DIMENSIONING; DIMSTYLES**

A screen menu with the following alternatives are shown:

SAVE

Saves a DIMSTYLE.

RESTORE
Restores a DIMSTYLE.

VARIABLE
List the values of all AutoCAD's dimensioning variables.

OVERRIDE
Temporarily changes dimensioning variable for a dimension and/or a DIMSTYLE.

RESET
Resets the default values for all dimension variables and sets or installs the DIMSTYLE MEKISO. Note that this function saves the default value in DIMSTYLE MEKISO. If you want to permanently change the DIMSTYLE, you can create your own. For further information about this procedure, see the AutoCAD manual.

Direct (FORMAT)

Insertion of form. Pre-set values are used to specify what form shall be chosen. The settings are done in the drawing's configuration before this command is selected.

Screen menu: **FORMAT; Direct**

More information:
See "Forms" on page 54.

DMin (FORMAT)

The function updates the title block with information from the .DMI file.

The program can be configured so that it knows the name of the block that contains the title block. This makes it unnecessary to select the block manually.

Screen menu: **FORMAT; DMin**

Digitizer menu: **FORMAT; DMIN**

Work procedure:

1. If necessary, point at the drawing head.

DMut (FORMAT)

The function updates the DMI file with information from the title block.

The program can be configured so that it knows the name of the block that contains the drawing's title block. This makes it unnecessary to point out the block.

Screen menu: **FORMAT; DMut**

Digitizer menu: **FORMAT; DMut**

Work procedure:

1. If necessary, select the title block.

DPARALL. (DIMENSION)

Gives a diameter dimension with the dimension line parallel to the specified start points of the extension lines. This is used, for example, to measure a diameter in a view that is not a circle or circle arc, for example a shaft.

Screen menu: **DIMENSION; DPARALL.**

Digitizer menu: **DIMENSIONING; Dparallell**

Work procedure:

1. Select a start point for the first extension line or press ENTER to choose a circle or circle arc.
2. **If start point has been given:**

   Select start point for the second extension line.

If ENTER has been pressed:

Select a circle. The dimension line will be drawn parallel to the tangent of the point where the circle was selected.

Draft->Design

This function checks if objects are in the “draft” layer and, if you wish, moves them to the connection-giving layer.

Digitizer menu: **CHANGE & MOVE**

More information:

1. Select the objects to be checked/changed. The objects, which are “draft”, will be marked.
2. Cancel or select the objects to be changed. ENTER changes the objects selected in step 1.

Ed_Copy

This function is similar to the AutoCAD command **Copy**, but this function also repairs/joins lines under the symbol that has been copied.

Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Select part of the diagram to copy.
2. Specify the base point or “M” for Multiple (compare with **Copy Multiple**).
3. Specify new insertion point(s).
4. If the selection contains symbols with designations, it is possible to edit the values of these. Enter “Y” (yes) and give the new values.

Ed_Copy + Edit

This has the same function as **Copy**, but this function also makes it possible to edit the item and terminal designations.

Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Select part of the diagram to copy.
2. Specify the base point or “M” for Multiple (compare **with Copy Multiple**).
3. Specify new insertion point(s).
4. If selected symbols have values, then a prompt will ask if these are to be edited. If a new value is to be given, then answer “Y” (yes) and specify the new value.

Ed_Erase

This function is similar to AutoCAD’s command **ERASE**, but this function repairs/joins the lines, which remain after a symbol is removed.

Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Select part of the diagram to remove.

Edit data (FORMAT)

This function opens a dialog box used to edit data in the drawing's title block.

Screen menu: **FORMAT; Edit data**

Note: The settings in the drawing's configuration are used to automatically search for the title block.

More information:
See "Forms" on page 54.

Edit dimensioning

Changes the dimension text of an existing dimension.

Screen menu: **DIMENSION; EDIT**

Digitizer menu: **DIMENSIONIN; Edit Dimension**

Work procedure:
1. Select the dimension for which the text should be changed.
2. Enter new text.

More information:
See "Dimensioning" page 91, for a description of how dimension text should be selected.

Edit dir. (MLPLOT)

Gives you the possibility to administrate the directory with the plotter configuration.

Screen menu: **MLPLOT; Edit.dir**

Work procedure:
1. To create a new directory, click **New**. To remove a directory, click **Remove**. To unlock a plotter directory, which another user in the network has locked, click on the button **Unlock**. This button can only be used if you are certain another user has locked the plotter directory.
2. Save your work by clicking on the button **Save data**.

More information:

See “Plotting” page 98.

Ed_Move

This function is similar to AutoCAD’s command **MOVE**, but this command joins and breaks the line under the symbol that is moved.

Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Select part of the diagram you want to move.
2. Enter the base point.
3. Specify the new insertion point.

EPS

Net/sheet

This is the first step in the EPS-generator (nets per sheet). This step analyses the sheet and searches for errors.

Work procedure:

1. Save the drawing.
2. Select the command.
3. Wait until the sheet has been analyzed.

Show error

Show all errors that have co-ordinates, which differ from zero, for example, an error that can be attributed to a position in the diagram. The error is then indicated by a cross and “error” text in the diagram. the function can show one error at a time or all at once. The secondary command, **error list**, lists all the errors on the screen, even errors that cannot be attributed to a co-ordinate.

Work procedure:

1. Activate the command and select the alternative that is prompted by entering the letter, which is upper case.

Error list/sheet

This lists all errors on the screen that the function **Electrical points/sheet** generates.

Show Status

Show all sheets in current EPS-database, and which status each sheet have. The status for each sheet is defined with the function **Net/sheet**, and whether there are error in the drawing or not.

Net/diagram

This creates an EPS-file with information from all the drawings in the EPS-database.

Work procedure:

1. Select the command.
2. Specify the EPS-file's search path and name.

Error list/diagram

This lists all errors on the screen that the function **Electrical points/diagram** generates.

ELP/erase

This deletes the drawing's information from the EPS-database.

Work procedure:

1. Select the command.
2. Wait until the sheet has been analyzed.

EXTEND (EDIT)

Extends a line a specified distance (or shortens it if a negative value has been given). The function can also extend both ends of a line simultaneously.

Screen menu: **EDIT; EXTEND**

Digitizer menu: **EDIT; EXTEND**

Work procedure:

1. Select which line should be extended, or press ENTER to extend both ends simultaneously.
2. Specify how long each end should be extended.

More information:
See Edit page 94.

EXTEND2 (EDIT)

Extends (or shortens) a line to the intersection point of a visible or hidden line. This command is a mixture of AutoCAD's Trim- and Extend- (for lines) and chooses the alternative that best suits a particular situation.

Screen menu: **EDIT; EXTEND2**

Digitizer menu: **EDIT; EXTEND2**

Work procedure:

1. Point at the line to be extended (or shortened). Point close to the end point to be extended.
2. Specify the start point for the desired boundary line, or press ENTER to choose an existing boundary line.
3. Point at the second end point of the desired line. (Only if an existing line has not been chosen previously by pressing ENTER).

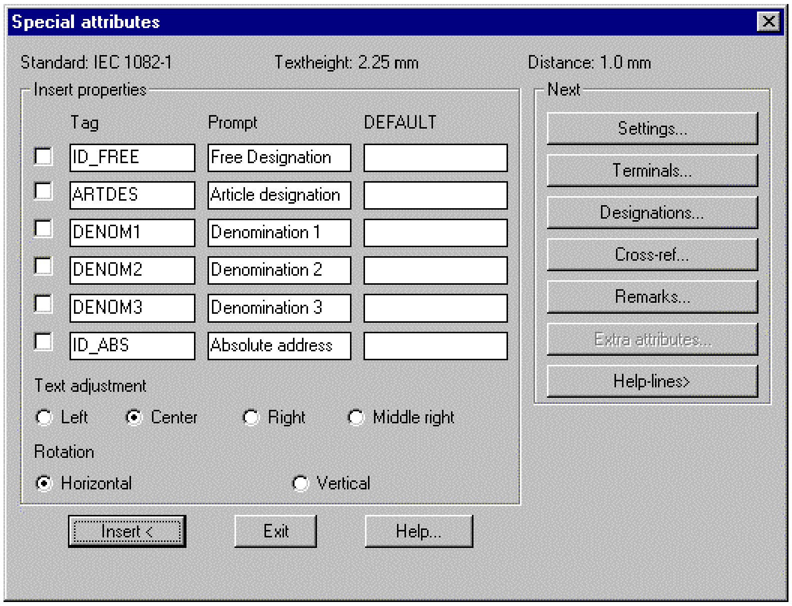
Note: The existing line that the chosen line should be extended or shortened to does not necessarily have to pass through the point of intersection. The function calculates the extension of the boundary line.

More information:

See “Edit” page 94.

Extra attribute

With the help of this dialog box you can create and insert your own attribute which you set-up when creating a new symbol.



Special attributes

Standard: IEC 1082-1 Textheight: 2.25 mm Distance: 1.0 mm

Insert properties

	Tag	Prompt	DEFAULT
☐	ID_FREE	Free Designation	
☐	ARTDES	Article designation	
☐	DENOM1	Denomination 1	
☐	DENOM2	Denomination 2	
☐	DENOM3	Denomination 3	
☐	ID_ABS	Absolute address	

Text adjustment

○ Left ● Center ○ Right ○ Middle right

Rotation

● Horizontal ○ Vertical

Figure 88. Special attributes

TAG PROMPT DEFAULT

This value can be changed to your own values. The values, which were found first in the dialog box, are kept listed in the file ELINIT.LSP in the MSC-directory.

Insertion properties

The attributes, which are to be inserted, are selected by a cross mark at the beginning of every row. At least one cross must be marked before an attribute is inserted. If more than one attribute is marked, then they will be inserted from top to bottom. Select the required text type and rotation.

Insert

Inserts all remark attributes.

Filter Select

Used to select symbols in standard reports. It filters (removes) symbol frames and insertion crosses.

Frames

The boundary frame can be four or more sided. However, it should have an even amount of sides e.g. 4, 6, 8 etc.

Digitizer menu: **BOUNDARY FRAMES**

A frame with four sides:

1. Select the required boundary frame.
2. Point at the insertion point, which is the top right hand corner.
3. When prompted “Bottom right hand corner/to point”, specify the bottom left hand corner.
4. Specify location designation.

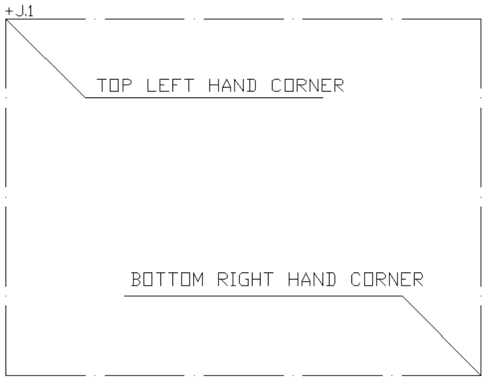


Figure 89. A frame with four sides

A frame with more than four sides:

1. Select the required boundary frame.
2. Point at the insertion point, which is the top right hand corner.
3. When prompted “Bottom right hand corner/to point”, specify a point with the same X- or Y- co-ordinate as the first point. Continue to draw the complete boundary frame by pointing at the corner, alternatively enter “C” for close (which will close the box), or “U” for undo (which will delete the last line drawn).
4. Specify location designation.

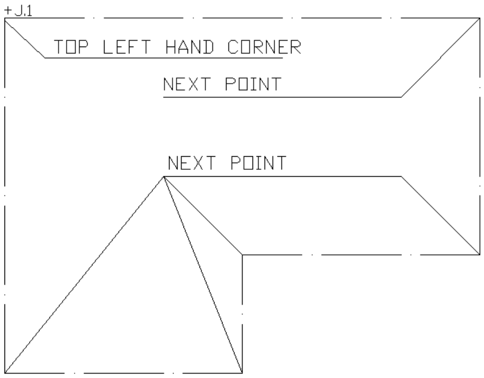


Figure 90. A frame with more than four sides.

Free conn.line

This function is used to draw a connection line. Every corner point must be pointed out manually. Otherwise, the functions for breaking lines etc. occur automatically.

Digitizer menu: **CONN.LINES.**

Work procedure:

1. Point at the start point.
2. Point at the end point.
3. Press ENTER after the last end point.

Free Symbol

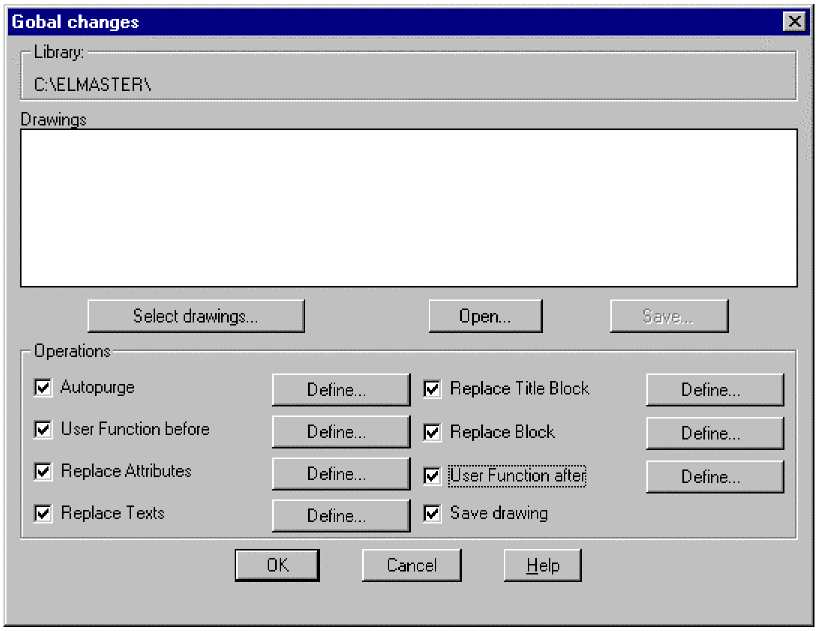
Draws a free symbol in the shape of a circle or rectangle. The function creates a so-called anonymous block. This should be handled as an ordinary block, but does not have a specific name.

The adjustments for type, form, measure and attribute are always saved from the previous session.

Measurements can be given in millimeters, inches or as a number of module units, for example, 19, 5S, 2C, and so on. Width and height may also be given by two corner points.

Global changes

With this function the dialog box in Figure 91 will appear. From here you can do changes that are global for a number of different drawings.



Gobal changes

Library:
C:\ELMASTER\

Drawings

Operations

☑ Autopurge
☑ User Function before
☑ Replace Attributes
☑ Replace Texts
☑ Replace Title Block
☑ Replace Block
☑ User Function after
☑ Save drawing

Figure 91. Global changes dialog window

Select drawings

This function activates a dialog box where you can choose the drawing you want to operate on. You can even sort these drawings in alphabetical order or alphabetical and numerical order.

Work procedure:

1. Start the function
2. Choose the drawings you want to operate on.

3. Finish with **OK**, **Cancel** or **Process all**. **OK** saves the drawings, which are marked for processing. **Cancel** ends the operation without saving. **Process all** saves all drawings for processing.

The drawings to be changed must be in the same directory as the drawings you are working with (this is usually the case because normally you will work with "project directory"). The dialog presents the drawings, which are found in the present directory. These drawings can be sorted in two different ways, either in alphabetical order or alphabetical and numerical order.

You can mark all the drawings, some of the drawings or you can unmark them. There is though a limit in AutoCAD which means that only up to 87 drawings can be marked. In the case where more drawings are to be processed, you can either click the button **Process all,** which will select all drawings, or you can specify drawings in a text file (ref).

Note: If a reference to a menu file, a font file or similar occurs in a drawing, which AutoCAD cannot find, the function cannot continue. Therefore, make sure that no such reference occurs in any of the drawings to be processed.

Example files

The MSC directory contains some example files that can be used together with global changes.

RITHUV3.TBL

Contains the TAGS from the drawing frame RITHUV3.DWG that is delivered with *el*Master.

FUNK.LSP

Simple example of a possible Lisp file "user function".

MLPLOT.LSP

Is an example of how plotting can be done with this function.

PLOT-directory contains a number of PCP-files for certain plotters including CalComp CCL600 for A4 and A3, CalComp TechJet A1, HP LaserJet II and Canon LBP 8IIR.

Note: The PCP-file a3l_a4l reduces an A3L-drawing to an A4L in exact scale. This means that you can take this type of plotting and magnify it on a copying machine to A3 with the correct scale.

Autopurge

This function removes a defined block definition from a drawing.

Work procedure:

1. Select **Autopurge; Define**
2. Select **Autopurge; level**
3. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

This operation can be repeated several times (levels) to remove blocks that are within each other, which can occur when you select more than one level.

Note: This function does not remove layers, text styles etc.

Change attributes

This function, replaces/changes the attributes of a block.

Work procedure:

1. Select Replace **attributes; Define**
2. Select **Add from block** or **Add**.
3. Select **Change** to change attribute value.
4. Select **Erase** to erase a block from a list.
5. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function without saving.

In order to be able to change attribute values, one has to know the block name, TAG, FROM and TO values (similar to the AutoCAD command ATTEDIT). Since it may be difficult to know which TAG's exists in a block and the block name, the function **Add from block** has been created. This function reads all attribute definitions from an existing block (TAG, PROMPT and VALUE) and the block name. From a list one can select one or more of there attribute definitions.

Note: Only the attribute TAG and VALUE will later be used. The PROMPT is only an aid to choose the right attribute. One can also choose weather to use the block name or if the selected attribute definitions are to be used with all blocks. In the latter case the block name is shown as a "*".

The function **Change** is used to change a marked line in the list, for example when one wants to alter the TO-value. It is also possible to specify if you want to add a part of the FROM-value.

Example: Imagine you have a FROM-value "Red skies". If you want to change the word "Red" in the FROM-value to "Blue", you use the command **Replace part of text**, then write "Red" in the FROM-value and write "Blue" in the TO-value. The word "Red" in the FROM-value will then be replaced with the word "Blue". The result in the attribute value will be that you have the words "Blue skies". If you had used the command **Replace whole text**, the attribute value would not have been changed.

Add
Adds a new line to the end of the list.

Remove
Removes a marked line.

Replace title block

Replaces a drawing's title block with another.

The block, which is to be inserted, must be found in AutoCAD's search path. If this is not the case, the complete search path to the block must be specified. To not lose any information from the old title block, the information is transferred to the new title block. If there are differences between the old and the new title block, or if you want to move a value from one attribute to another, you can with the help of the transfer file specify how you want these values transferred. The transfer file is a common text file with two columns. In the left-hand column the TAG's from the old title block are specified. In the right hand column the equivalent TAG for the new title block is specified. The file should always have the extension TBL. If no transfer file is specified, the TAG's will be identical in the new and old title blocks.

The function even updates the configuration information.

Work procedure:

1. Select **Change frame/block; Define**
2. Select **Add from block** or **Add**.
3. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

Replace texts

This function replaces/changes text.

Work procedure:

1. Select **Replace text; Define**
2. Select **Add from text** or **Add**.
3. Select **Change** to change the text.
4. Select **Remove** to remove text from a list.
5. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

To be able to replace text, you must first know the FROM- and TO-values. To avoid having to write in the text, you can use the function **Add from text<** to register a drawing's existing text. The function **Change** is used to change a marked row in a list, for example, when you want to change the TO-value. You can even specify if you want to change a part of the text string.

Example: Imagine you have a FROM-value “Half moon”. If you want to change the word “Half” in the FROM-value to “Full”, you use the command **Replace part of text**, then write “Half” in the FROM-value and write “Full” in the TO-value. The word “Half” in the FROM-value will then be replaced with the word “Full”. The result in the attribute value will be that you have the words “Full moon”. If you had used the command **Replace whole text**, the attribute value would not of been changed.

Add
Ads a new line to the end of the list.

Remove
Removes a marked line.

Replace block

Replaces an existing block with another. Information in attributes can be transferred in the same manner as in the function **Change frame/block**, for example with or without transfer file.

Work procedure:

1. Select **Replace block; Define**
2. Select **Add from block** or **Add**
3. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

User function before/after

Function to replace a title block.

Work procedure:

1. Select **User function before/after; Define**
2. Select the LISP-function
3. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

The user is enabled to create a function written in AutoLISP. The Lisp file should be written so that the function is performed when the file has been loaded (similar to the AutoLISP function **LOAD**).

Save drawing

This function is used to save a drawing after processing.

If this alternative is not chosen, the drawing, which you have processed, will not be saved with the changes you have specified. This is used mainly with "batch"-plotting, when you want, for example, to set-in the correct layer definition preceded by it's own function (see MLLAYER.LSP) and “batch”-plotter with it's own function after (see MLPLOT.LSP). When this alternative is not chosen, the layer settings you have specified will not be saved and the settings, which were found in the drawing before the processing, will be kept.

Save

This function saves selected drawings.

Work procedure:

1. Select the text file where the drawing shall be saved or create a new text file.
2. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

Note: This function can only be selected if you have chosen drawings from **Select drawing**.

Open

This function opens a text file to register a drawing.

Work procedure:

1. Select the text file where the drawing, which is to be processed, is listed.
2. Select **OK** or **Cancel**. **OK** saves the drawing in the text file whilst **Cancel** ends the function, without saving.

In this text file, the drawings, which are to be processed, must be specified. The drawings must be described with their complete search path but without the file type (extension). Every row in the file describes a drawing.

Note: If a reference to a menu file, font file or similar occurs in any drawing, which AutoCAD cannot find, the function cannot be continued. Therefore, make sure there are none of these references in the drawings, which are to be processed.

GRID

Sets values for the variable GRID. The usage of GRID is described in the AutoCAD manual.

Screen menu: **MLKONFIG; GRID**

Work procedure:

1. Select required setting of GRID from the screen menu. GRID can be transparent.

Hbase line (DIMENSION)

Dimensions according to the method for superimposed running dimensioning. Horizontal base line.

Screen menu: **DIMENSION; Hbase line**

Digitizer menu: **DIMENSIONING; Hbase line**

Work procedure:

1. Select start point of the extension line for the dimension origin. If an Hbase line was the last thing to be drawn, it is possible to continue drawing by pressing ENTER when prompted for the first extension line and then pointing at the extension line for the dimension origin.
2. Specify the second extension line.
3. Place the dimension line.
4. Edit the dimension text.
5. Give start point for the next extension line and edit the dimension text. Repeat this procedure until all the desired dimensions have been drawn.
6. Press ENTER to quit the command.

Note: Extension lines can be specified on either side of the first extension line. This can be done alternately.

The program can be configured either for printing the text parallel to the dimension line or perpendicular to it.

More information:
See “Dimensioning” page 91.

Help-lines

Insert
Inserts a frame around attribute definitions and text. The function is used to achieve the correct insertion point for other attribute definitions in relation to those within a frame. The frame is inserted in the designers layer, which must be switched on if the frame is to be visible.

Delete
Deletes all inserted help lines.

HELPLINE (EDIT)

Draws a horizontal or vertical help line that passes through a selected point and stretches to the drawing's borders (LIMITS).

Screen menu: **EDIT; HELPLINE**

Digitizer menu: **EDIT; HELPLINE**

Work procedure:

1. Select the point the line is to pass through. Instead of selecting a point, you may also press ENTER to change between horizontal and vertical orientation of the line.

More information:
See “Edit” page 94.

Hole Punch

With this dialog box you can create and insert hole punch attributes when you create new symbols.

Default values

Type a reference to a symbol for a hole punch or retrieve an existing hole punch from the Sym.Lib.

Insert

Inserts all remark attributes.

HOMETEXT (DIMENSION)

Restores one or several dimension texts to the position specified through the DIM variable settings.

Screen menu: **DIMENSION; HOMETEXT**

Digitizer menu: **DIMENSION; HOMETEXT**

Work procedure:

1. Select one or several texts to be restored.

More information:

See “Dimensioning” page 91.

Horizontal (DIMENSION)

Makes a linear dimension with a horizontal dimension line.

Screen menu: **DIMENSION; Horizontal**

Digitizer menu: **DIMENSIONING; Horizontal**

Work procedure:

1. Select start point for the first extension line or press ENTER to choose a line.
2. **If the start point was given above:**

   Select start point for the second extension line.

   If ENTER was chosen:
   Select line to be dimensioned.
3. Place the dimension line.
4. Edit the dimension text.

More information:
See “Dimensioning” page 91.

Increment item designation

Increment item designation. Used by the functions **Copy Row** and **Copy One**.

On the question **Increment for item designation**, answer with a positive integer or by pushing return to accept the default value.

Initiate blockbuild

Initiates the blockbuild function and resets the variables in the drawing environment so that they are adjusted to the blockbuild functions.

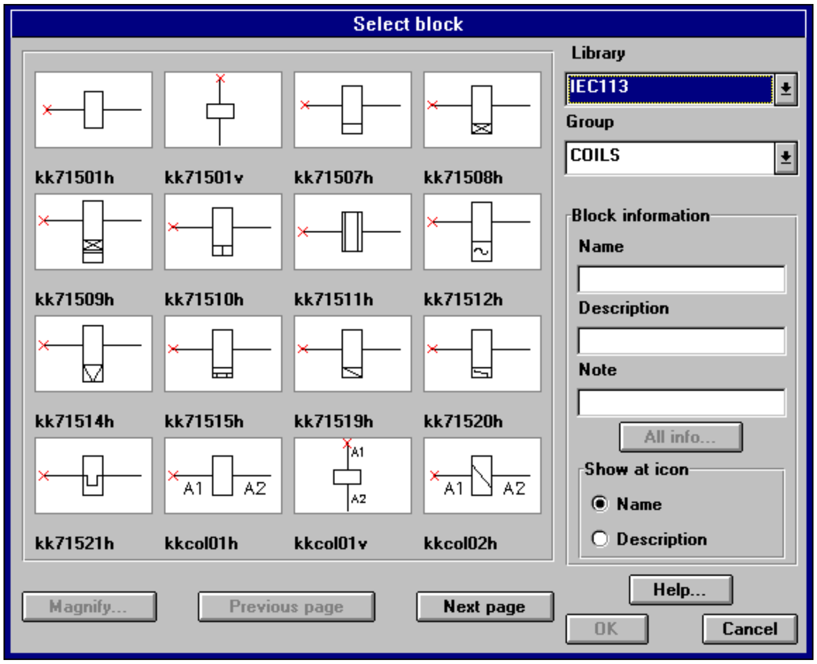
Insert (BLOCKLIB)

Inserts a block from a chosen symbol library (directory). The block is inserted in different ways depending on the directory settings.

Screen menu: **BLOCKLIB; Insert**

Digitizer menu: **BLOCKLIB; Insert**

Figure 92 shows a dialog box for inserting a block from BLOCKLIB. The dialog box have different appearances and functions depending on the type of symbol that should be inserted.



Library: IEC113

Group: COILS

Show at icon: ● Name ○ Description

Figure 92. Select block dialog window

Library and group

The symbols are divided into groups. These groups are in separate directories in the operative system. In the uppermost right corner, in the icon menu, you can see which group the various symbols belong to. If you want, you can change groups here. However, normally you will select the group from the pull-down menu.

Block information

Here you will find the block name for the symbol and it’s description. The description is complimentary information about every individual symbol. This describes the symbol in more detail than the block name.

For each separate symbol a note could also exist.

All information

Show all the parameters that are tied to the marked symbol. Here you can find information about the symbol that you are about to insert, including for example the scale-factor. You can also see which category (if one has been specified) the symbol belongs to.

Show at icon

Here you can choose if the block name or the description of the symbol should be shown in the picture.

Next page/Previous page

These buttons you use to see if there are more pages with symbols.

Magnify

Enlarges the marked symbol.

Work procedure:

1. Select a directory from the directory list.
2. Select a group from the list.
3. Select a block from those shown (point at the required block). When you have chosen a block, the information about the block is shown.
4. Press **OK** or double-click on respective icon to insert the block.
5. Specify insertion point.

A dialog box appears. From here you handle the symbol's item designation, terminal designation and other data connected to it. This information can also be changed afterwards with the function **Text/Value**.

If the dialog box does not appear this is due to a setting in the configuration. In Attribute handling you specify if attribute question should be given and if they should be stated in a dialog box or in the prompt.

Remark: When creating your own symbols the value on **Symbol type** will control which dialog that is selected. This value is stored in the block library and in the symbol after it has been inserted. The value is fetched from the field CATEGORY2 in block database. The value can be edited with the function **Symb.Cat**.

More information:
In the chapter “Create your own block”, you can read more about how you change the parameters of a symbol or build your own symbols.

Insert (FORMAT)

Shows a dialog box for the insertion of format.

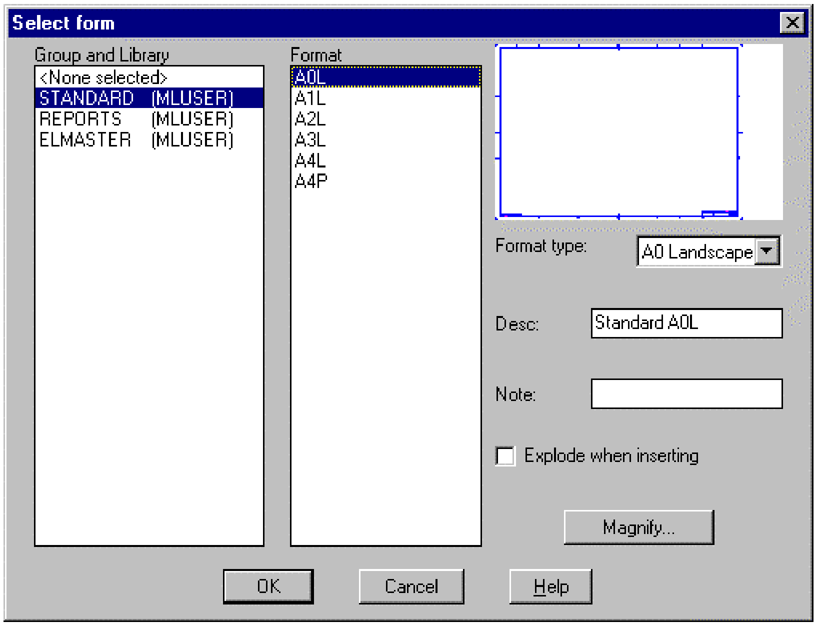
Screen menu: **FORMAT; Insert**

Work procedure:

1. Choose which set-up (directory) the form should be selected from (only if there are many form set-ups available).
2. Choose what form should be inserted.
3. Press **OK**.

The directory, group and block name is saved in the drawing's configuration to make it possible for you to find the revision list block that belongs to the form.

The title block's block name is saved in the drawing's configuration so that you should be able to update the title block automatically with the information from the drawing archive. Even values like the form's reference net is transferred to the drawing.



Group and Library
<None selected>
STANDARD (MLUSER)
REPORTS (MLUSER)
ELMASTER (MLUSER)

Format
A0L
A1L
A2L
A3L
A4L
A4P

Format type: A0 Landscape

Desc: Standard A0L

Note:

☐ Explode when inserting

Figure 93. Select form

Format type:
Shows which format type that belongs to the form.

Desc.:
Name of the form.

Note:
Note about the form.

Explode when inserting:
Here you choose whether the block should be exploded during insertion or not.

More information:
See “Forms” page 54.

Inserting symbols from the symbol library

Inserts symbol from the chosen library.

The insertion method depends on the settings for each symbol library. It is possible, for example, to choose whether all the insertions should be made using a constant angle or whether a different angle should be used for each insertion.

The insertion process also depends on the **Insertion of hole punch** configuration.

To install the hole punch, the symbol hole punch reference (the PUNCH field) must be marked and the corresponding symbol must be in the library HOLES.

Insertion of hole punch

Inserts a hole punch.

For this function to be performed, the hole punch reference must be marked. The value of the hole punch reference must be the same as the symbol name (file name) of the hole punch. The hole punch should be in the symbol library HOLES.

INTERSECT BREAK (EDIT)

Breaks up all elements located in the selected intersection point.

Note: the polyline with an end point in the selected area cannot be broken up.

Screen menu: **EDIT; INTERSECT BREAK**

Digitizer menu: **EDIT; INTERSECT BREAK**

Work procedure:

1. Select intersect point (the object snap INTERSECT is activated automatically).

More information:

See “Edit” page 94.

ISOSELES TRIANGLE

Draws a triangle according to the specified base and height of the triangle. The height must be specified in accordance with the insertion point.

Screen menu: **DRAWBOX; ISOSELES**

Digitizer menu: **DRAWBOX; Box 6**

Work procedure:

1. Enter insertion point.
2. Enter the base.
3. Enter the height.
4. Enter the angle.

JOIN (EDIT)

Connects two lines to one.

Screen menu: **EDIT; JOIN**

Digitizer menu: **EDIT; JOIN**

Work procedure:

1. Point out the first line (near the end point, which is to be moved).
2. Point out the second line (near the end point, which is to be moved).

A new line will now be drawn. The start point will be the furthest endpoint on the first line and the finish point the furthest endpoint on the second line. In this case it means the end point, which has the larger distance to the point where the line was selected.

More information:
See “Edit” page 94.

LARGE RADIUS (EDIT)

Connects two lines to an arc where the angle of the arc is 180 degrees or more. Compare AutoCAD's Fillet where the angle of the arc should be less than 180 degrees. The function also handles parallel lines.

Screen menu: **EDIT; LARGE RADIUS**

Digitizer menu: **EDIT; LARGE RADIUS**

1. Select the lines.
2. Where applicable, select the radius.
3. **If the lines are parallel and do not lay in line:**
   Select the center point of the arc.

   If the lines are parallel and lay in line:
   Select the side where the arc is to be drawn.

More information:
See “Edit” page 94.

Last symbol

Inserts the last chosen block.

Screen menu: **BLOCKLIB; Last.blk**

Digitizer menu: **BLOCKLIB; Last block**

Work procedure:

1. Insert the block according to the command sequence, depending on the directories setting.

More information:

See “Symbols” page 63.

Last group

Inserts a block from the last selected symbol library (directory). If no symbol library has been selected, the function prompts for a library. The block is inserted in different ways depending on the library settings.

Screen menu: **BLOCKLIB; Last.grp**

Digitizer menu: **BLOCKLIB; Last Library**

Work procedure:

1. Select block by marking the name in the list or by pointing at the icon.
2. Insert the block according to the command sequence as specified in the library settings.

More information:

See “Symbols” page 63.

LAYER

A function that changes the status of one or several layers. The layers can be turned on, turned off, thawed, frozen, locked or unlocked. You can also change to another active layer.

?

Gives a list of existing layers in the drawing and their properties.

Screen menu: **LAYER; ?**

Work procedure:

1. Select the layer you want to be displayed.

Mark

Marks all the objects that are in a chosen layer by using HIGHLIGHT.

Screen menu: **LAYER; Mark**

Work procedure:

1. Specify the name of the layer that is to be marked.

All On, All Off

Switches on or off all layers in the drawing.

Digitizer menu: **LAGER; All On or All Off**

Dialog

Opens a dialog box that offers the same command choices as the screen menu.

Digitizer menu: **LAYER; DIALOGUE**

Work procedure:

1. Select the desired command in the dialog box.

ON, OFF, FREEZE, THAW

Changes the status of a selected layer. The layer can either be selected directly or chosen from the plate. The alternatives **Select** and **the whole group** on the plate can be used.

Digitizer menu: **LAYER; (ON, OFF, FREEZE, THAW)**

Work procedure:

1. Select the layers for which the status should be changed. The desired layer can be chosen by selecting it from the digitizer menu.

Delete

Deletes all the objects that are in a chosen layer.

Screen menu: **LAYER; Delete**

Work procedure:

1. Specify the name of the layer, which is to be deleted.

ON, OFF, THAW, FREEZE, LOCK, UNLOCK

This function affects the status of the selected layer

Screen menu: **LAYER; (ON, OFF, THAW, FREEZE, LOCK, UNLOCK)**

Digitizer menu: **LAYER; (On, Off, Thaw, Freeze)**

Work procedure:

1. Activate the respective function.
2. Select the layer where status is to be changed.

Select

Switches from one active layer to another.

Screen menu: **LAYER; Select**

Digitizer menu: **LAYER; Select**

Work procedure:

1. Select name of the layer that is to become the new active layer.

Note: You can either choose a layer name from a list of the layers that belong to the active layer group, write in a name, or specify the layer name by pointing at an object placed on the desired layer.

Layer off

You can “turn off” a layer with this command.

Digitizer menu: **LINE TYPE/LAYER/COLOR**

Select from object<
With this command you can mark an existing object (for example a line) and find out which layer it is in.

More information:
See “Layers” page 56.

Layer on

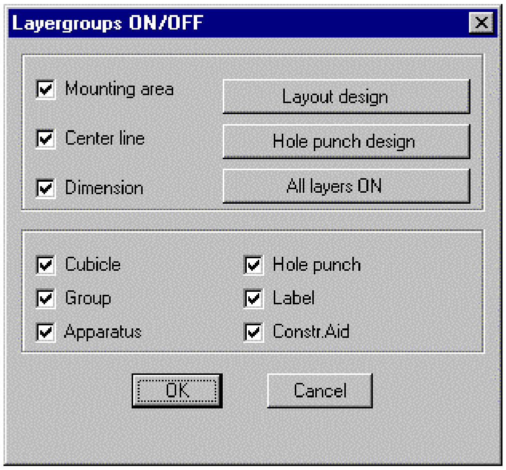
You can “turn on” a layer with this command.

Digitizer menu: **LINE TYPE/LAYER/COLOR**

More information:
See “Layers” page 56.

Layer Status

Toggles layers and layer groups on and off.



☑ Mounting area
☑ Center line
☑ Dimension

☑ Cubicle
☑ Group
☑ Apparatus
☑ Hole punch
☑ Label
☑ Constr.Aid

Figure 94. Layer groups ON/OFF

Information in the dialog box shows the status of the different layers/layer groups. Selections marked with a cross are on.

The dialog box also makes it possible to activate the layer to create layout or hole punch drawings, as well as to activate all the layers.

The **Mounting area**, **Center line** and **Measurement adjustment selections**, turn on and off the corresponding layers in all the layer groups.

The layer groups used by the program are listed in the chapter and should not be altered.

LAYER LEVEL

?

Gives a list of existing layers, including properties, for a selected level.

Screen menu: **LGRUPP; LEVEL; ?**

Work procedure:

1. Select layer level to be displayed.

LIST

Gives a list of existing layer levels in the drawing.

Screen menu: **LGRUPP; LEVEL; LIST**

HIGHLIGHT

Highlights all objects on a layer, which belong to the layer level selected by highlighting (HIGHLIGHT).

Screen menu: **LGROUP; LEVEL; Highlight**

Work procedure:

1. Select the layer level that shall be highlighted.

ON, OFF, THAW, FREEZE, LOCK, UNLOCK

These functions influence the status of layers belonging to the layer level to be selected.

Screen menu: **LGROUP; LEVEL; (ON, OFF, THAW, FREEZE, LOCK, UNLOCK)**

Work procedure:

1. Activate each function.
2. Select the layer level for which the status should be changed.

Length Bar

Changes length of a bar, for example mounting bar, terminal bar and so on. This function is also useful for cable channels (conduits).

Bars and cable channels (conduits) are drawn as so-called 1 x 1 blocks, for example they have the correct height but their length is only 1 mm. To change the length, the block’s X-scale must be changed to the desired length.

The length can be specified in any of the following ways:

- By specifying a terminal point when the system prompts, **Point out length or ENTER for the dialog box**.
- Via the dialog box, by specifying two points in the drawing, by pointing at another object in the drawing or, by specifying a measurement in millimeters, inches or a number of module units.

The length can be a positive or negative number. If the number is negative, the symbol is drawn in the opposite direction of the original.

Level

Changes the level of the object. The level of the object co-ordinate in Z direction is used to remove the underlying hidden lines with the command **HIDE**.

The present level is shown in the dialog box. If several objects have been chosen and these are not on the same level, the text **varied** appears in the bottom section of the dialog box.

A new level can be specified in different ways: either by choosing one of the “0-6” buttons pointing to another object or by manually entering an integer.

When the level is chosen with the “1-6” buttons, a help text appears in the bottom section of the dialog box that describes the purpose of each level.

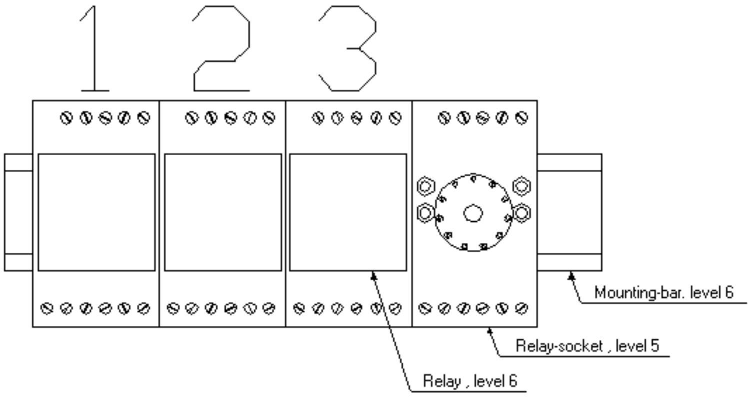


Figure 95. An object with different levels.

LEXPLODE (EDIT)

Explodes a block so that the part in the block maintains the properties (layer, line type and color) that they had prior to the explosion of the block. It is possible to choose whether the values of attributes should be retained or not.

Screen menu: **EDIT; EXPLODE**

Digitizer menu: **EDIT; LEXPLODE**

Work procedure:

1. Select the block to be exploded.

More information:

See “Editing” on page 94.

LGROUP

Functions that change the status of a layer group. The layer group can be turned on, turned off, thawed, frozen, locked or unlocked. You can also change to another active layer group or create new layer groups.

?

Gives a list of the layers which are in a selected layer group and the properties which it has

Screen menu: **LGROUP; ?**

Work procedure:

1. Select the layer group that you want to be displayed.

LIST

Gives a list of the layer groups, including a description of each, contained in the drawing.

Screen menu: **LGROUP; LIST**

Highlight

Highlights objects on a selected layer group by marking with HIGHLIGHT.

Screen menu: **LGROUP; Highlight**

Work procedure:

1. Select the layer group to be highlighted.

NEW DESCRIPTION

Changes the description line for an existing layer group.

Screen menu: **LGROUP; NEW DES**

Work procedure:

1. Select the layer group for which the description should be changed.
2. Enter the new description line.

CREATE NEW

Creates a new layer group with an accompanying description line

Screen menu: **LGROUP; CREATE NEW**

Digitizer menu: **LGROUP; CREATE NEW**

Work procedure:

1. Select name for new layer.
2. Give a description of the layer group (one row).

ON, OFF, THAW, FREEZE, LOCK, UNLOCK

These functions influence the status of the layer that belongs to the selected layer group.

Screen menu: **(ON, OFF, THAW, FREEZE, LOCK, UNLOCK)**

Digitizer menu: **LGROUP; (ON, OFF)**

Work procedure:

1. Activate the respective function.
2. Select the layer group for which the status should be changed.

Select

Switches from one active layer group to another. If you have configured the system for automatic layer locking, the layer group you switch to becomes unlocked and the layer you leave becomes locked.

Screen menu: **LGROUP; Select**

Digitizer menu: **LGROUP; Select**

Work procedure:

1. Select the layer group that will become the new active layer group.

Note: You can either choose a layer name from the list of layers that belong to the active layer group, or by pointing to an object belonging to the desired layer group.

Look-up apparatus

Remark: This function is not a general function in *el*Master design, an additional product is needed.

If the drawing is connected to a project database, this function gives you the possibility to look-up a designation for an existing apparatus in the database.

Mark

When wire dimensions or other data are to be linked to a conductor, this is usually done by drawing a line and a stroke on those wires. This is a function to simply move these strokes and a line out to the text.

Digitizer menu: **CABLE/CORD SPEC.**

Work procedure:

1. Point at the start point.
2. Point at the applicable conductor, which the stroke will be inserted on, horizontally or vertically from the start point.

Mark layer

You can use this command to see what a specified layer contains. The layer which you “mark” can then be edited by selecting **previous** under the command **select object**.

Digitizer menu: **LINE TYPE/LAYER/COLOR**

Select from object<
With this command you can mark an existing object (for example a line) and find out which layer it is in.

More information:
See “Layers” page 56.

Mechanical link

Draws a mechanical link between, for instance, a coil and a contact with an attached description. The first point must be a “receptive” connection point in which there are places on the contacts and coils where a mechanical link is possible. If the start point or end point is on another link, the function creates a coupling point. The important thing for connection-table generation, if used, is that the link should lay on top of the symbol so that it is given the correct item designation. Hence, it is not necessary to point to or stop at a contact that is included, but the line should be drawn across the symbol.

Undo can be used from the screen menu to undo the last line segment.

Digitizer menu: **CONN.LINES**

Work procedure:

1. Draw the mechanical link by pointing at the required start point, end and also corner.

MIDDLE POINT LINE

Draws a line in which the middle point and length are specified.

Screen menu: **DRAWBOX; MIDLINE**

Digitizer menu: **DRAWBOX; Box 8**

1. Enter insertion point.
2. Enter the line's length.
3. Enter the line's angle.

Mleader (DIMENSION)

Makes a leader line.

Screen menu: **DIMENSION; Mleader**

Digitizer menu: **DIMENSIONING; Mleader**

Work procedure:

1. Select start point for the leader (insertion of arrow) or press return to choose a circle or circle arc.
2. Select the positioning of the leader by pointing at the point it should pass through. The drawn segment can be removed with UNDO. Quit with ENTER.
3. Write in the desired text or write VERTICAL to print the text vertically.
4. If several text lines are desired, select the first or press ENTER to quit.

More information:

See "Dimensioning" page 91.

MLPLOT

Functions that order a print-out of the drawing which presently is edited to one of the printers which AutoCAD is configured for

Screen menu: **MLPLOT**

Digitizer menu: **FORMAT; MLPLOT**

Work procedure:

1. Decide if the plotting is to be made with hidden line removal using the command **Hide On/Off**.
2. Select scale and appearance from the screen menu.
3. Select which plotter should perform the printing.
4. If necessary, select which setting (PCP file) should be used if this hasn't already been done (FIT only).

More information:

See "Plotting" page 98.

Alternatives selecting scale factor and which part to plot

Scale factor:

SCALE	Prints out the drawing with a pre-selected scale factor.
FIT	Prints out the drawing as large as possible. Proper scale factor not used.

Section:

LIMITS (or L)	Selects a detail of the drawing that lays inside AutoCAD's LIMITS.
DISPLAY	Selects to include everything visible on the screen.
EXTENTS (or E)	Selects the smallest detail of the drawing possible in which everything is still included.
VIEW	Selects to include everything visible in a named view.
WINDOW	Selects to include everything visible in a specified window.

Move text 1 mm

With this command you can move a text row 1 mm upwards.

Select: Point at the text row that is to be moved.

Last: Selection is made with the method “Last”, for example the last inserted object.

Digitizer menu: **CHANGE & MOVE**

Work procedure:

1. Press the window SELECT. Point at the text row.
2. Press the window LAST.

Mult. conn.line

Creates a number of parallel connection lines in the x-y or y-x direction. The connection lines will have a fixed distance between them at the start point and the same or another fixed separation distance and at the end. The lines may be separated in both the x- and y-direction, for example the start points of two parallel lines have different x and y co-ordinates.

If the x and y co-ordinates of both the start points and the end points differ, the direction of the connection will be oriented in the x-direction from the start point.

If the start or end point is on another connection, a connection point is automatically created

Digitizer menu: **CONN.LINES**

Work procedure:

1. Point at the start point.
2. Specify the pitch ,in the start point, by pointing out the distance and the direction.
3. Point at the end point.

4. Specify the pitch, in the end point, by pointing out the distance and the direction.
5. Enter the number of conductors in the multiple connection.

Mult.select Article

Assigns articles to the selected symbols.

Remark: This function is not a general function in *el*Master design, an additional product is needed.

Mult.select Change designations

Changes the designations on one or several symbols. With this function you can change the designations on a complete terminal row. The benefit with this function is that it also changes designations that are not visible and that all designations are changed with one command. This function can also be used on other apparatuses/symbols and then gives them all the same designation.

Digitizer menu: **EDIT DESIGNATIONS**

Work procedure:

1. Select the symbols that you want to change.
2. Give the new designation.
3. Verify the change for every symbol.

Mult.select Change references

Changes the sheet reference on a row of references or other symbols.

Digitizer menu: EDIT DESIGNATIONS

Work procedure:

1. Select the references that you want to change.
2. Give a new sheet reference.
3. Verify all the references that you want to change.

Mult.select Symbol-Category type

Changes the symbol type and/or the symbol category for the selected symbols. The function searches the block database for correct values.

More information:

See chapter "Categories and types".

Mult.select Terminals

Changes the terminal designations on a row of symbols, for example, terminal blocks.

Work procedure:

1. Select the symbols to be changed. Select the symbols in the order that you want to have them numbered in.
2. Give the start value.
3. Give the enumeration factor (integer).
4. Give new sheet designation.
5. Verify the changes for all terminals.

Mult.select Object category

Changes the object category on the selected symbols.

Remark: This function is not a general function in *el*Master design, an additional product is needed.

NEWTEXT (DIMENSION)

Replaces the dimension text of one or several dimensions.

Screen menu: **DIMENSION; NEWTEXT**

Digitizer menu: **DIMENSIONING; NEWTEXT**

Work procedure:

1. Select new text for the chosen dimension. Press ENTER to choose an estimated dimension value.
2. Choose one or several dimensions that should receive the new text.

More information:
See “Dimensioning” page 91.

Non scale (DIMENSION)

Draws a line under the dimension text to indicate that this dimension is non-scaling. This is used, for example, to dimension a shaft with a break.

Screen menu: **DIMENSION; NON SCALE**

Digitizer menu: **DIMENSIONING; Non-scaling dimension**

Work procedure:

1. Point at the dimension.

More information:

See “Dimensioning” page 91.

Objiram (BLOCKLIB)

Object in frames ('Objiram) is a transparent function, which is used to select an object. Note that this function does not work on frames or insertion points (these are filtered away). The function can be used to do operations, as for example CHPROP in a block in more than one frame at a time

Screen menu: **BLOCKLIB; Objiram.**

Work procedure:

1. Start an editing command (e.g. MOVE, ROTATE, etc.).
2. Use this function to select the objects you want to edit. (including frames and insertion points). The function automatically filters frames and insertion points away from the chosen object.
3. Continue with the command you chose in step 1.

OBLIQUE (DIMENSION)

Changes the angle of the extension line for one or several dimensions.

Screen menu: **DIMENSION; OBLIQUE**

Digitizer menu: **DIMENSIONING; OBLIQUE**

Work procedure:

1. Specify a slanting angle.
2. Select one or several dimensions to slant.

More information:
See “Dimensioning” page 91.

One conn.line

Unlike the **free connection** command, you only have to point at the start and end points. The connection line continues in the same direction as the connection point of a symbol or the end point of a connection line.

The difference between “normal” connections and “main circuits” is the width of the line. Normal lines have a width dimension of 0.0 mm whilst the main circuit lines are 0.5 mm.

Note: The function can not chose direction in all cases. When the start point is in an intersection, the function can not decide in what direction the new line should be drawn.

Digitizer menu: **CONN.LINES**

Work procedure:

1. Point at the start point.
2. Point at the end point.

More information:
Se “Connection lines” page 80.

On/Off (MLPLOT)

Sets whether it should be possible for the plotter to use hidden lines removal.

Screen menu: **MLPLOT; On/Off**

Digitizer menu: **FORMAT; MLPLOT; On/Off**

Note: This setting should be made before the plotting is started. It does not work together with AutoCAD's ordinary Plot command.

More information:
See “Plotting” page 98.

One reference

Insert the required remark.

Digitizer menu: REFERENCES

Work procedure:

1. Point at the insertion point, for example an end point on a connection line.
2. Specify the value for Signal/Reference.
3. Specify the value’s reference.

ORIGIN

Functions to move and restore the origin, for example move UCS-origin.

MOVE

Moves the origin (UCS).

Screen menu: **ORIGIN; MOVE**

Work procedure:

1. Select new origin.
2. Select angle.

RESTORE

Restores the origin (restores UCS to WORLD).

Screen menu: **ORIGIN; RESTORE**

Parallel (DIMENSION)

Makes a line dimension with a dimension line parallel to an imagined line through the specified extension line's start point.

Screen menu: **DIMENSION; Parallel**

Digitizer menu: **DIMENSIONING; Parallel**

Work procedure:

1. Select start point for the first extension line or press ENTER to choose a line.
2. **If the start point was given above:**

   Select start point for the second extension line.

   If ENTER was chosen:
   Select line to be dimensioned.
3. Place the dimension line.
4. Edit the dimension text.

More information:
See "Dimensioning" page 91.

PARALLEL (EDIT)

Draws a line parallel to a reference line.

Screen menu: **EDIT; PARALL**

Digitizer menu: **EDIT; PARALL**

Work procedure:

1. Select reference line.
2. Select start point for the new line.

3. Select length of the new line. If the distance is pointed out on the screen, the angle toward the start point is not considered, only the marked distance.

More information:

See “Edit” page 94.

PICKBOX (MLCONFIG)

Sets the values for the variable PICKBOX. The usage of PICKBOX is described in the AutoCAD manual.

Screen menu: **MLCONFIG; PICKBOX.**

Work procedure:

1. Select new values for the variable PICKBOX.

PLOT (MLPLOT)

AutoCAD's command for plotting. You can choose whether or not to work with a dialog box with the variable CMDDIA.

Screen menu: **MLPLOT; PLOT**

Digitizer menu: **FORMAT; MLPLOT; PLOT**

Work procedure:

1. Set desired value of the variable CMDDIA.
2. Activate the command.

Note: For further information, see the AutoCAD manual. Note that the settings can be saved in a configuration file (PCP file).

More information:

See “Plotting” page 98.

Properties

Each entity created in AutoCAD, independent of type, has properties. This functions handles the properties layer, line type and color.

Lgroup

Move a number of chosen entities to another layer. The entities that are moved may belong to different layer groups.

Screen menu: **PROPERTIES; LGROUP**

Digitizer menu: **PROPERTIES; LGROUP**

Work procedure:

1. Select the objects that are to be moved to another layer group.
2. Select the layer to receive the entities. The alternative **Select from entity** gives the possibility of choosing an entity to take the layer group from.

Changes line type for a number of chosen entities. You can specify the name of the line type to change to, or choose the alternatives **Near** or **Thin**. These alternatives work well with the standard AutoCAD line types.

Layer

Moves a number of chosen entities to another layer. The entities that are moved may belong to different layers.

Screen menu: **PROPERTIES; LAYER**

Digitizer menu: **PROPERTIES; LAYER**

Work procedure:

1. Select the entities to be moved to another layer.
2. Select the layer to receive the entities. The alternative **Select from entity** gives the possibility of choosing an entity to take the layer from.

Line type

Screen menu: **PROPERTIES; LINE TYPE**

Digitizer menu: **PROPERTIES; LINE TYPE**

Work procedure:

1. Select the entities that should receive a new line type.
2. Select new line type. The alternatives **Near** and **Thin** choose a new line type if one exists.

Color

Changes color of a number of chosen objects.

Screen menu: **PROPERTY; COLOR**

Digitizer menu: **PROPERTY; COLOR**

Work procedure:

1. Select the objects that are to be given a new color.
2. Select new color.

Note: The colors of the dimensions are controlled by special variables. For further information see the AutoCAD manual.

Rad1 (DIMENSION)

Makes a radius dimensioning with the dimension line drawn from the center. The arrow will be in contact with the point where the arc/circle was chosen.

Screen menu: **DIMENSION; Rad1**

Digitizer menu: **DIMENSIONING; Rad1**

Work procedure:

1. Mark the arc or circle that should be dimensioned.

More information:
See “Dimensioning” page 91.

Rad2 (DIMENSION)

Makes a radius dimensioning with the dimension line drawn from the center and extended an arbitrary distance.

Screen menu: **DIMENSION; Rad2**

Digitizer menu: **DIMENSION; Rad2**

Work procedure:

1. Mark the arc or circle that should be dimensioned.
2. Edit the dimension text.
3. Point out the length of the leader.

Note: With the AutoCAD variable DIMTOH it is possible to decide whether the leader should extend a horizontal “tongue” for the dimension text to be printed on. Use the SETVAR command to change this setting.

DIMTOH = 0 The “tongue” will not be drawn.

DIMTOH = 1 The “tongue” will be drawn.

More information:
See “Dimensioning” page 91.

RECTANG. (DRAWBOX)

Draws a rectangle according to the specified diagonal corner points.

Screen menu: **DRAWBOX; RECTANG.**

Digitizer menu: **DRAWBOX; Box 1**

Work procedure:

1. Enter the first corner.
2. Enter the second corner.
3. Enter the angle.

MIDRECT. (DRAWBOX)

Draws a rectangle according to the specified middle point of one side of the rectangle together with the width and height.

Screen menu: **DRAWBOX; MIDRECT**

Digitizer menu: **DRAWBOX; Box 3**

Work procedure:

1. Enter insertion point.
2. Enter the rectangle’s width.
3. Enter the rectangle’s height.
4. Enter the angle.

REFPOINT

This function is used to save a point and use the saved point.

USE

Refers to a saved point according to another function’s request.

Screen menu: **REFPOINT; Use**

SAVE

Saves the co-ordinates for a specified point.

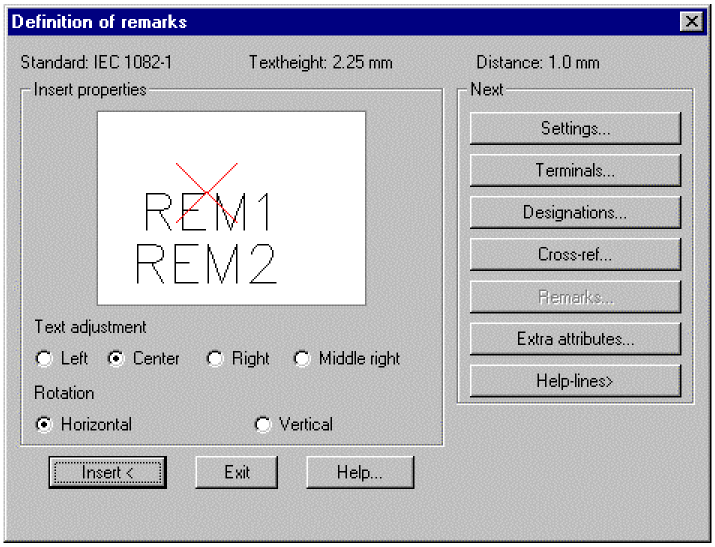
Screen menu: **REFPOINT; SAVE**

Work procedure:

1. Select point.

Remarks

With help from this dialog box, you can insert remark attributes (REM1-REM2) when you create a new symbol.



Definition of remarks

Standard: IEC 1082-1 Textheight: 2.25 mm Distance: 1.0 mm

Insert properties

Text adjustment

○ Left ● Center ○ Right ○ Middle right

Rotation

● Horizontal ○ Vertical

Figure 96. Definition of remarks

Insertion properties

Select the required text type and rotation.

Insert

Insert all remark attributes.

Remarks on/off

Switches on/off remarks about a symbol. The function switches off remarks that are “on” and switches on remarks that are “off”.

To switch of a single remark, use the function **Attribute Layer**.

Digitizer menu: **EDIT DESIGNATIONS.**

Work procedure:

1. Select one or more symbols.

Note: The remarks are not deleted, the function only changes the layer.

Remove DB-link

Removes the symbol’s link to the project database.

Replace (FORMAT)

Changes the present form and replaces it with a new form. The information in the title block can be saved and transferred to the new drawing's title block. You can specify a transfer file, which describes the transfer of the TAG-name, if the new drawing's title block has a different TAG-name. The transfer file is an ASCII-file with two columns where the first gives the TAG-name in the old drawing's title block and the other TAG-name is the new drawing's title block.

Screen menu: **FORMAT; Replace**

More information:

See “Forms” page 54.

Reset blockbuild

Restores the variables in the drawing environment that the initiating function changes.

REVISION

MARKING

Inserts a revision mark in the drawing.

Screen menu: **REVISION; MARKING**

Digitizer menu: **FORMAT; REVISION; MARKING**

Work procedure:

1. Select insertion point for the marking.
2. Select revision numbers.

Note: The block name can be changed in the configuration of the drawing.

LIST ROW

Inserts a row in the revision list.

Screen menu: **REVISION; LIST ROW**

Digitizer menu: **FORMAT; REVISION; LIST ROW**

Work procedure:

1. Fill in the values in the dialog box.

Note: Block name, number of rows, insertion point and distance between rows can be altered in the configuration of the drawing.

Revision Cloud

Draws a so-called revision cloud in the drawing.

1. Specify the revision number or the designation or press ENTER for the present revision. If the drawing is connected to *el*Master project or another document handling system that handles revisions, revision designations will be fetched from this system.

   For every new revision designation a new layer is created.
2. Specify what the cloud should look like by pointing out points in the drawing.
3. Specify the insertion point for the marker.

Note: the block name can be altered in the configuration of the drawing

RHOMB (DRAWBOX)

Draws a rhomb according to the specified diagonal and one side.

Screen menu: **DRAWBOX; RHOMB**

Digitizer menu: **DRAWBOX; Box 4**

Work procedure:

1. Enter the insertion point.
2. Point at the diagonal.
3. Enter one side.
4. Enter the angle.

RIGHTAN. (DRAWBOX)

Draws a right angle triangle according to the specified base and height of the triangle. The height is specified in accordance with the insertion point.

Screen menu: **DRAWBOX; RIGHTAN**

Digitizer menu: **DRAWBOX; Box 7**

Work procedure:

1. Enter insertion point.
2. Enter the base.
3. Enter the height.
4. Enter the angle.

ROTATED (DIMENSION)

Makes a line dimension with a dimension line drawn perpendicular to the extension lines. The angle of the dimension line is specified separately.

Screen menu: **DIMENSION; ROTATED**

Digitizer menu: **DIMENSIONING; Rotated**

Work procedure:

1. Select start point for the first extension line or press ENTER to choose a line.
2. **If the start point was given above:**

   Select start point for the second extension line.

   If ENTER was chosen:
   Select line to be dimensioned.
3. Place the dimension line.
4. Edit the dimension text.

More information:
See “Dimensioning” page 91.

Save to database

This function is only available when the drawing is connected to a valid database. The function analyses the drawing, saves the designations of the symbols and other information to the database. It also assigns an index to the symbols so that the designations later on can be updated from the database with the function **Update from database**.

Remark: This function is not a general function in *el*Master design, an additional product is needed.

Select layer

This alternative is used when you want to select another layer.

Digitizer menu: **LINE TYPE/LAYER/COLOR**

Select from object<
With this command you can mark an existing object (for example a line) and find out which layer it is in.

More information:
See “Layers” page 56.

SET ini

This is for setting the different search paths. You can set the search path to the program code as well as to the various MLUSER-trees.

Screen menu: **MLCONFIG; SET ini**

More information:
See chapter “Installation Guide”.

SETVAR

Changes or displays the current settings of AutoCAD's system variables.

Screen menu: **MLCONFIG; SETVAR**

Work procedure:

1. Choose the required system variable and select a new value, or select “?” to receive a list of current values.

Set text height

Sets the text height of “draft” text. The default height is always the height last used in the drawing, or, for a new drawing, 2.5.

Digitizer menu: **DECORATIVE TEXT.**

Work procedure:

1. Point to the box with the selected text height on the application menu or the screen menu.

Settings blockbuild

This is a dialog box for the setting of the parameters, which are used for the blockbuild functions.

Standard

Control of terminal designations to be positioned over connection lines (IEC1082) or (IEC113). These are the only two alternatives available.

Text height terminal

Determines the text height for terminals. The values are 2.25 not 2.5. This is because of the thickness of the text, normally 0.25 mm.

Text height other

Determines the text height of the other text in symbols, for example, location designations and cross-references.

Distance to line/text

Determines the distance between inserted attributes and their insertion points you point at. Example: You have chosen IEC 113-standard, which means that the terminal designation should be positioned under the connecting line. In this case, 1 mm means that the distance between the connection line and the top part of the terminal designation is 1 mm.

Force to layer "0"

If this alternative is marked with a cross, it means that the graphical details (which you have drawn or created from **Primitives**) will be found in layer 0. Thus, the symbol will acquire the properties to choose color and line type according to the layer it is found in. But, this will only occur if colors and line type are set to **Bylayer** in your drawing

Force to color "Byblock"

Means that the graphical details will adopt the color of the block. If you select this variant, you can then later change the color of the symbols by use of the command **color**.

Force to linetype "Byblock"

Means that the graphical details will adopt the line type of the block, which you have set.

Show database error

Show all possible errors that may appear in the functions **Update from database** and **Save to database**.

Show DB-link

This function shows all the symbols that have a link to the database.

Show scale factor (FORMAT)

This command shows the drawing’s scale factor.

Screen menu: **FORMAT; SCALE ; Show**

SNAP

Sets the values of the variable SNAP. The usage of SNAP is described in the AutoCAD manual.

Screen menu: **MLCONFIG; SNAP**

Spec. Cond.

From here you can create a “draft” description.

Symbols
Choose between different “draft” symbols to insert on the connection

Number of wires
Specifies the number of wires in a connection.

Arrow
Inserts a “draft” arrow to specify the connection where there is room.

Mark
Inserts a “draft” mark to specify the connection where there is room.

Supply

3-Phase, Vertical, Horizontal
Used to quickly draw 3-Phase, supply, vertical and horizontal phase-lines with references. The alternative **More** gives access to all supplies in the symbol group SUPPLY.

TEDIT (DIMENSION)

Changes the position of the dimension text and the insertion angle of a dimension.

Screen menu: **DIMENSION; TEDIT**

Screen menu: **DIMENSIONING; TEDIT**

Work procedure:

1. Mark the dimension to be edited.
2. If the dimension text should be moved, point out a new middle point for the text or choose one of the following alternatives from the screen menu:

- **Left**
  The dimension text is left justified.

- **Right**

  The dimension text is right justified

- **Home**

  Restores the dimension text to the specified position specified through the DIM variable setting.

- **Angle**

  Rotates the dimension text in an arbitrary angle relative to the dimension line.

More information:
See “Dimensioning” page 91.

Terminals

With the help of this dialog box, you can create and insert terminal attributes with the applicable connection point when you create a new symbol.

Connection points are important in that they enable information to be passed on to other systems, based on the logic net the circuit diagram describes.

Text positioning
Select the required text positioning. During insertion, the terminal designation will have the following text adjustment.

Horizontal symbol:

Left for a terminal above a line and with the outlet to the right.

Right for a terminal above a line and with the outlet to the left.

Top/Left for a terminal under a line and with the outlet to the left.

Top/Right for a terminal under a line and with the outlet to the right.

Vertical symbols:

Left for a terminal to the left and with the outlet upwards.

Right for a terminal to the left and with the outlet downwards.

Top/Left for a terminal to the right and with the outlet upwards.

Top/Right for a terminal to the right and with the outlet downwards.

These text adjustments will give the same distance to the line, even if the text height is changed.

Order number

This number couples together a terminal designation and a connection point. A symbol must have a unique number. The number is automatically counted when an insertion occurs.

Terminal code

This text is used to code the differences between the various terminals in a symbol. The default value is ”A”. Other values can include, for example, “I” for internal side and “E” for external side etc.

Default value

The default value (pre-selected value) for terminal designations.

Insert

Insert terminal designations and connection points at the specified insertion point

Term. blocks

Designation and numbering

With repeated insertions of terminal blocks it is possible to count-up the terminal blocks numbers. For this function, it is necessary to set the parameters that are going to apply for this count-up.

Digitizer menu: **TERM.BLOCKS**

Reply to the prompts or press ENTER to accept the default value. The default values are:

- Start number terminal block number <1>
- Count-up factor <1>
- Question terminal block number <N>

Insertion one

Insertion of terminal block in the drawing. The terminal type must be decided according to “Type” on page 252.

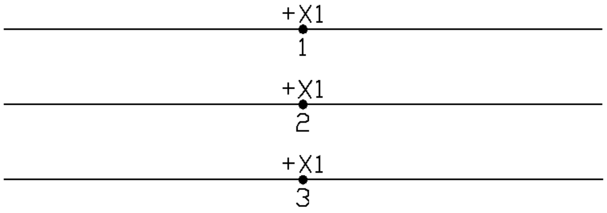
When a terminal block is inserted in the drawing, the system suggests a value for the terminal designation. If you enter your own value, which is a mixture of letters and numbers, then the value will only be counted-up if the it ends with a number. For example, the next value after “A6” will be “A7”.

Digitizer menu: **TERM.BLOCKS.**

Work procedure:

1. Point at the insertion point.
2. Specify the value for the designation and the terminal block number.

Insertion row



Inserts a complete row of terminal blocks, either in the X-direction or in the Y-direction. All terminal blocks will get the same designation as the one given to the first block. The item designation is switched off automatically on all terminal blocks except the first.

Input of designation and number is done in the same manner as described above, under **Insertion**

Digitizer menu: **TERM.BLOCKS.**

1. Point at the insertion point for the first terminal block.
2. Point at the insertion point for the second terminal block.
3. Give the number if terminal blocks that you want to insert.

Insertion of terminal blocks on connections

Item designation and terminal designation works in the same way as insertion of a row.

Digitizer menu: **TERM.BLOCKS.**

Work procedure:

1. Point at the connections that you want to insert terminal blocks on or specify a line by selecting on two points, terminal blocks will then be inserted in the crossing between the line and the connections. These two ways of insertion can be combined.
2. Finish with ENTER.

Manual insertion

Insertion of a terminal block. Same function as in **insertion one**, but without the terminal block count-up capability.

Digitizer menu: **TERM.BLOCKS.**

1. Point at the insertion point.
2. Specify a value for the location designation and terminal block number.

Type

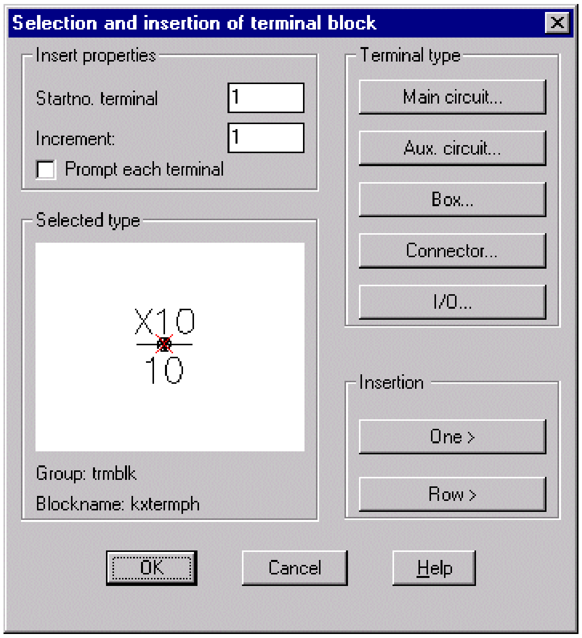
In this dialog box the parameters, which are used when inserting terminal blocks, are controlled. Insertion of both single and rows of terminal blocks is also possible.

Terminal types used in boundary frames are placed 2.5 mm inside the frame (1 “snap”-step) and have text justified 1 mm from the frame. The terminal's 2-side is turned outwards.

Digitizer menu: **TERMINAL BLOCK**

Work procedure:

1. Select type of terminal block symbol.
2. Specify the value of the start number and the count-up factor and also the setting **Prompt each terminal**.
3. Select the insertion alternative or press **OK** to save the setting.



Selection and insertion of terminal block

Insert properties

Startno. terminal: 1

Increment: 1

☐ Prompt each terminal

Selected type

Group: trmblk

Blockname: kxtermph

Figure 97. Selection and insertion of terminal block

Insert properties

The start number of the terminal and count-up factor decides the value of the terminal block's number.

Prompt each terminal

If this is marked, the function for inserting the terminal block row will ask for every terminal block's number.

Terminal block type:
With these buttons you can enter Blocklib. In Blocklib you can select which type of terminal or connection block you want to insert.

Although the circuit diagram is the base for the connection table, it is important that you select the correct type because there are differences, i.e. 1- and 2- side (internal- and respectively external side).

Main circuits use the symbols in the TRMMAIN group
Auxiliary circuits use the symbols in the TRMBLK group
Box use the symbols in the TRMBOX
Ansl.don use the symbols in the CONNECT
I/O use the symbols in the I_O_UNIT

Insertion:

One>
Insert the selected terminal block. Same function as insertion.

Row>
Insert a row of terminal blocks. Same function as insertion.

TEXT

Functions used for creating, editing and altering the appearance of the text. There are also functions used for reading in and printing out text to and from external files.

WIDTH

Changes the width factor for a number of texts.

Screen menu: **TEXT; Edit; Width**

Work procedure:

1. Select texts for which the width factor should be changed
2. Specify the new width factor.

REPLACE

Replaces one word with another in a number of texts.

Screen menu: **TEXT; Edit; Replace**

Work procedure:

1. Specify the texts for which the replacement is to occur.
2. Specify the old text.
3. Specify the new text.

DDEDIT

Edits text. For further information, see the AutoCAD manual.

Screen menu: **TEXT; Edit; DDEDIT**

Work procedure:

1. Select a text to edit.

HEIGHT

Changes text height for a number of texts.

Screen menu: **TEXT; Edit; Height**

Work procedure:

1. Select the texts in which the height should be changed
2. Select **new text height** from the screen menu.

The alternative FREE allows the user to specify text heights that do not exist on the screen menu.

OBLIQUE ANGLE

Changes oblique angle for a number of texts.

Screen menu: **TEXT; TEXT; DTEXT; Oblique**

Work procedure:

1. Select texts for which the oblique angle should be changed.
2. Specify new oblique angle.

READ

Reads an ASCII-text from a file and writes it to the drawing.

It is possible to decide how the text should be written, for example: start point (Center/Middle/Right), text rotation angle and distance between the lines, the first line of text to read, the number of lines to read in, if all lines are to be underlined, if all lines should be overstruck, if the letters are to be in the upper- or lower-case, if a column setting is to be performed.

Screen menu: **TEXT; Edit; Read**

Work procedure:

1. Select file name on the file from which the text is to be read.
2. Give the start point or Center/Middle/Right.
3. Give rotation angle.
4. Choose whether to change text options.

These are:

- Specify distance between lines.
- First lines to be read in.
- Number of lines to be read in.
- Underlining of all lines.
- Overstriking of all lines.
- Conversion of all letters Upper-case/Lower-case/No.
- Divide the text in columns.
- Distance between columns.
- Number of lines per column.

WRITE

Printouts selected texts to a text file, which can then be edited by an editor.

Screen menu: **TEXT; Edit; WRITE**

Work procedure:

1. Mark the text that is to be printed out on the file.
2. Specify the file name of the text that is to be printed out.

STYLE

This function’s sub alternatives are shown on a screen menu. The alternatives are as follows:

STYLE
Choose another text style (STYLE).

LIST
Lists all text styles defined in the drawing.

Text heights from 1.8 - 10.0.
This sets current height of text to a specified value. Note that the text height compensates for pen width. A text height of 5.0 mm is equal to 4.5 mm in AutoCAD units.

Free
Sets the current text height to 0. This means that you will be prompted for a text height when inserting text.

Screen menu: **TEXT; style**

Work procedure:

1. Select desired function from the screen menu.
2. Select desired alternative shown on the screen menu.

TEXT/DTEXT

These are the two main functions for typing in text. They have a screen menu by which AutoCAD’s different text options can be set.

The height of the text is adjusted so that when plotting with the proper pen, the text height will be correct. For example, for 3.5 mm text, the height is set to 3.15, for example the height minus the pen width.

Screen menu: **TEXT; TEXT** or **DTEXT**

Digitizer menu: **TEXT; TEXT** or **DTEXT**.

Work procedure:

1. If no option should be set, the start point is specified for the text.
2. Select angle for the text.

Note: For further information about the commands **Text** and **Dtext**, as well as all options shown on the screen menu, see the AutoCAD manual.

Text height terminals

This function is used to temporarily change the text height if numerous symbols or a complete diagram need to be done.

The normal text height for diagrams, according to the IEC 617 standard, is 2.5 mm. 2.25 mm will become 2.5 mm if the drawing is plotted with the correct pen width (2.25 + 0.25 = 2.5). If the drawing is plotted on A4 format (scale 0.707), the text height 1.62 (1.62 + 0.18 = 1.8) is not approved for use according to the standard, but can be preferred for terminal designations, especially if the symbols in the diagram are congested.

Screen menu: **MSC.SET; Text height Terminal**

Work procedure:

1. Set the required text height in the configuration.
2. Select the symbols to be changed with the appropriate method

If the change is temporary, return the text height in the configuration to its original value.

Select

This command lets you point out the symbols that you want to change.

All

This command makes the selection with the method “All”.

Last

This command makes the selection with the method “Last”

Text Left, Centred etc.

Start the setting of the text adjustments as shown below:

Left
The text will be to the right of the start point.

Center
The first text row will be horizontally centered around the center point and vertically over the point.

Middle
The first text row will be horizontally positioned around the center point and vertically on the point.

Right
The text will be to the left of the start point.

Digitizer menu: **DECORATIVE TEXT**

Work procedure:

1. Point to the box with the selected centering.
2. Point to the insertion point.
3. Specify the direction of the text in degrees (ENTER gives 0 degrees).
4. Enter the text.
5. Finish with ENTER on a blank line.

Text/Value

This function is used to change text (for example designations for a symbol). You can only change the values of one object (symbol or text) at the time. When an object has been selected, a dialog box is shown with the object's values. The appearance of the dialog box depends on the type of object that has been selected. If the selected object is an *el*Master symbol (object type INSERT with x-data), the value of SYMBOLTYPE will control the appearance of the dialog box. Other objects than symbols may be changed with this function, i.e. text (object type TEXT or MTEXT), attribute definitions (object type ATTDEF) and dimensioning text (object type DIMENSION).

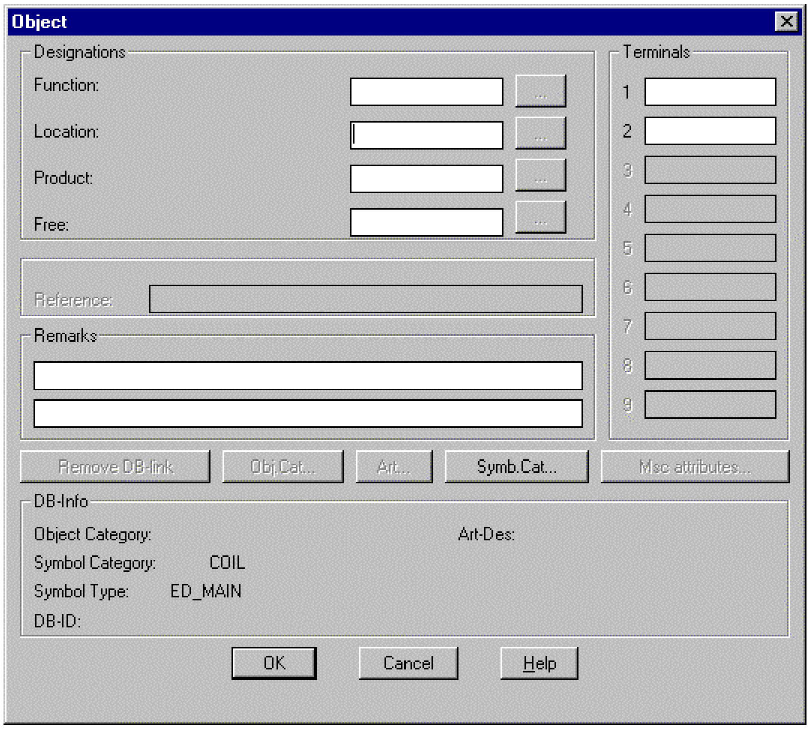
If the drawing is connected to a project database, you can look up existing apparatuses in the database.

The appearance of the dialog box depends on the selected object. There is a global dialog box for objects, and there are specially designed dialog boxes for frames, signals and cross-reference symbols.

Digitizer menu: **EDIT DESIGNATIONS.**

Work procedure:

1. Select symbol or text.
2. Change designation.



Symbol Category: COIL

Symbol Type: ED_MAIN

Figure 98. Object dialog window

Explanation of the dialog box:

Designations

Change designation in the text field or use the search button to find an existing designation in the project database.

Terminal

Change the terminal designation in the text field. You can only fill out as many designations (max. 9) as there are in the symbol.

Reference

Change reference (reference to page) in the text field. If the symbol lacks the reference field, this field will be off. By using the function for updating of cross-references, you can get this field filled out automatically.

Other attributes

If the symbol contains attributes that are not part of *el*Master standard, this button will be on and the values of these attributes will be accessible through an extra dialog.

Set object category

Searches the project database for object categories. The value in the symbol category is used to limit the selection of object categories. For more information about object categories, see also *el*Master project manual.

Set symbol category

Searches the block database for valid symbol categories and symbol types. See chapter “Categories and types”.

DB-info

Displays the information about the symbol that is used to determine the characteristics of the symbol in the project database.

Object category

The object category is changed with the **Set Obj.Cat** button (signals excluded). Frames can have three different categories for their different designations, because the designations can be different objects.

Article

Starting with a object category, you can search the article registry and assign an article to an apparatus. With the filter you can narrow down the number of articles in the list.

Symbol category, symbol type

The symbol category and the symbol type can be changed with the **Set Symb.Cat** button. Any change to these values might affect the functionality of the program. You can not change these values to anything you want, only to certain given values.

The symbol type is retrieved from the CATEGORY2 field in the symbol database and is used to decide which function that should be used in the drawing environment, for example, the dialog box used to change symbol values is defined by this value.

DB-link (DB-ID)
This value is an internal pointer to the symbol in the project database and is given to the symbol when it is saved, or in any other way connected, to the project database. If this value is missing, it means that the symbol is not connected to the project database. This value, and the connection to the project database, can be removed by using the **Remove DB-Link** button.

THEORET. (DIMENSION)

Draws a box around the dimension text to indicate that there is a Form- and Phase tolerance in connection with this dimension.

Screen menu: DIMENSION; THEORET.

Digitizer menu: DIMENSIONING; Theoretical

Work procedure:

1. Point at the dimension.

More information:
See “Dimensioning” page 91.

Total report

This report covers all the blocks that are in BLOCKLIB. For every block group, a drawing is created with those blocks that are included in the group. There is no requirement on the form.

Screen menu: **BLOCKLIB; Tot.rep.**

TRIANGLE CORNER POINTS

Draws an isosceles triangle according to two specified corner points in a rectangle.

Screen menu: **DRAWBOX; TRIANG**

Digitizer menu: **DRAWBOX; Box 5**

Work procedure:

1. Enter insertion point.
2. Enter the second corner point of the rectangle that is to register the triangle.
3. Enter the angle.

TROTATE (DIMENSION)

Changes the orientation of the dimension text for one or several dimensions. Dimensions that have the dimension text parallel to the dimension line will have the text rotated orthogonal towards the dimension line and vice versa.

Screen menu: **DIMENSION; TROTATE**

Digitizer menu: **DIMENSION; TROTATE**

Work procedure:

1. Mark all dimensions that should have their dimension text rotated.

More information:

See “Dimensioning” page 91.

Update crossref.

This function adds cross-references (sheet references) on signals and symbols that appear on several places in a drawing.

Remark: This function is not a general function in *el*Master design, an additional product is needed.

More information:
See “Cross-references” page 84.

UPDATE (DIMENSION)

Changes the appearance of one or several dimensions to the specifications indicated in the DIM variable settings. For information about the DIM variable, see the AutoCAD manual. UPDATE can also be used for restoring the look of a simple base line dimension after it has been edited.

Screen menu: **DIMENSION; UPDATE**

Digitizer menu: **DIMENSIONING; UPDATE**

Work procedure:

1. From the screen menu specify whether the object, to be updated, is a base line or ordinary dimensioning.

Base line
Mark all dimension details of the simple base line dimension

Ordinary
Mark all dimensions to be updated. If entities that are not dimensions are included in the choice, these will be ignored.

More information:
See “Dimensioning” page 91.

Update from database

Remark: This function is not a general function in *el*Master design, an additional product is needed.

This function is only available when the drawing is connected to a valid database. The function analyses the drawing and updates the designations and other information for the symbols that has been given an index by the function **Save to database**.

Vbase line (DIMENSION)

Dimensions according to the method for superimposed running dimensioning.

Screen menu: **DIMENSION; Vbase line**

Digitizer menu: **DIMENSIONING; Vbase line**

Work procedure:

1. Select start point of the extension line for the dimension origin.
2. If a “Vbase line” was the last thing drawn, it is possible to continue drawing by pressing ENTER when prompted for the first extension line and then pointing at the extension line for the dimension origin.
3. Specify the second extension line.
4. Place the dimension line.
5. Edit the dimension text.
6. Give a start point for the next extension line and edit the extension text. Repeat this procedure until all the desired dimensions have been drawn.
7. Press ENTER to quit the command.

Note: Extension lines can be specified on either side of the first extension line. This can be done alternately.

More information:

See “Dimensioning” page 91.

Vertical (DIMENSION)

Makes a line dimension with a vertical dimension line.

Screen menu: **DIMENSION; Vertical**

Digitizer menu: **DIMENSION; Vertical**

Work procedure:

1. Select start point for the first extension line or press ENTER to choose a line.
2. **If the start point was given above:**

   Select start point for the second extension line.

   If ENTER was chosen:
   Select line to be dimensioned.
3. Place the dimension line.
4. Edit the dimension text.

More information:
See “Dimensioning” page 91.

VIEWS

ADJUST

Adjusts views in **paper space** so that they fall into the right position in relationship to each other via a common point.

Screen menu: **FORMAT; VIEWS; Adjust**

Digitizer menu: **FORMAT; VIEWS; ADJUST**

Work procedure:

1. Select the view to be adjusted by pointing on the frame of the view.
2. Select reference view by pointing at the frame of the view.
3. Select the point that is to be located on the same "level" in x- or y-direction. The function now scans the view to be adjusted to its right position.

CREATE

Creates four views according to European “E” or American “A” view placing. The view in the bottom right corner is an ISO view.

Screen menu: **FORMAT; VIEWS; CREATE**

Digitizer menu: **FORMAT; VIEWS; CREATE**

Work procedure:

1. Change to **paper space** by setting the “setvar” variable TILEMODE to the value “0” (with the AutoCAD command TILEMODE).
2. Select whether the views are to be placed according to European “E” or American “A” standards.
3. Specify the insertion point of the method symbol.

Zero number

Insert a draft “zero numbering” in the diagram that is intended for the users who do not generate connection tables or who want to manually insert part number after the table generation.

This function can be used in one of two ways. If you are referencing a single net-section, for example one conductor, you point to the conductor. If there are multiple parts, you point to the connection points.

The function will ask for a start number every time it is used and will automatically increase the value each time you point to a line or every other time when you point to a connection point will automatically assign the same value to two connection point.

The user is also provided with the possibility to move the value since two values may occur in one connection points with more than one parts interconnecting.

Work procedure:

1. Specify if the number start with 0.
2. Enter part number.
3. Point to the line or connection.
4. Move the designation or continue with ENTER.
5. Start the next alternative or end.

Structure and Definitions

CHAPTER 9

To make your work run as smoothly as possible, it can be good to be acquainted with the following definitions.

Accessories
Are items that are needed for a device to be properly mounted and correctly operated. Examples include cable channels (conduits), relay sockets, mounting bars (DIN bars) and frameworks.

Arrangement drawing, hole punch design
A variation of layout design, which rather than showing the devices uses hole punches and illustrations of the mounting area to illustrate where devices should be mounted.

Arrangement drawing, layout design
Assembly drawing for one or more enclosures in which the placement and positioning of devices is shown.

Attribute
Most symbols have certain fields linked to them to specify, for example, terminal numbers and location designations. These fields are called the symbol's attributes.

Boundary frame
A boundary frame is used to specify that a group of symbols have the same location designation. Also used to specify a physical, mechanical or functional relation between parts. Refer to IEC 617-2, symbols 02-01-01 and 02-01-06.

Click

This means that you press and release the left-hand button on the mouse.

design layer

Whatever is to be analyzed by the connection table generator should be “design”, for example drawn on a “design” layer.

Connection point

This is used when a terminal needs to be specified, but when there is no finished symbol available. Is used often together with a complete boundary frame to create a so-called “Black box”.

Draft layer

When a circuit diagram is to be analyzed by the connection table generator, certain “draft” layers can be added to the layer if they are not intended to be analyzed by the table generator.

Default

Is a pre-set basic value or a previously set value, which is chosen automatically if you do not choose a particular value (just press ENTER) after a prompt by the system.

Device

Articles placed in an enclosure. Each device is represented as a function/item in the circuit diagram. Contactors, push buttons, fuses, etc. are all examples of devices.

Device group

A set of devices mounted on separate plates or within a separate framework. Examples include motor starters in low voltage switchgear, or relay protections mounted on a framework.

Directory

A directory is a specific section of the hard disk or diskette in which files can be placed. Directories are created using operating system commands or with the functions that are available for handling adjustments and symbol directories.

Enclosure

The physical unit in which devices are placed i.e. fuse box, cubicle, routing area etc. Contains a number of mounting planes and, possibly, support items such as plates, sockets, locks, etc.

ENTER

ENTER represents the return-key on the keyboard. Alternatively, you can press the second button of the mouse or the cursor of the digitizer.

Format type

Type of format. Examples of common format types: A3H (A3 horizontal) and A4V (A4 vertical). This format type is registered by the plot function.

Form

Name of a form definition. A form definition consists of information about format types and a description about how the form should be drawn up (drawing head, revision head, bookmark etc.). The form name needn't be the same as the format type.

Form setting

Several forms gathered in a group. Each group is equivalent to a directory in an MLUSER tree.

Hole pattern reference

Attribute to device symbol the value of which identifies the hole-punching pattern for the device.

Icons

Pictures in the digitizer menu, the pulls down menu (for example line type), dialog boxes or icon menus. These are used to emphasize a command in such cases where a picture or a graphic description is more suitable than words.

Initial drawing

When a drawing is first created, a ground plan is installed in which the drawing environment is already configured. It is possible, for example, to have a ground plan with a customized layout and ready-made drawings that are already installed. The ground plan is chosen during the configuration of AutoCAD (see installation instructions), and can also be specifically explained when starting a new drawing. The ground plan may sometimes be called the prototype drawing.

Mechanical link

This is used to specify a mechanical connection between two different symbols, for example, between a coil and a contact in a relay. Refer to IEC 617-2, symbol 02-12-01.

Module

Enclosures often have a certain spacing requirement for height and width. An example for this are “S-modules” which are 44.45 mm in height.

Mounting Area

For mounting plates in an enclosure, mounting area refers to the location where, for practical reasons, devices can be mounted (the crease of a metal plate, for example, would affect its mounting area). For devices, mounting area refers to the area needed to mount the device (for example, it is not possible to mount certain push-button close to each other).

Mounting plane

Is a surface which devices are mounted on. An enclosure may contain one or more mounting planes. Mounting planes may include one or more device groups.

Net

Is an apparatus terminal (connection points) with the same electrical potential which are linked to each other. In the drawing they are shown as lines which inter-connect terminals in different symbols. Note: the terminal symbol is an apparatus with two outlets that share the same net. A net must always consist of at least two connection points.

Point

When this request is made, point the mouse or digitizer cursor to the requested or desired location, or choose an object on the blueprint by moving the cursor into the drafting area; then press on the mouse button.

Reference net

A special coordinate system that is used when you want to refer to a specific part of the drawing.

Revision head
This part of a form is often placed in the bottom left corner of the drawing frame (it can also be placed in other positions).

SPACE
SPACE represents the space bar on the keyboard.

Terminal
Is a connection apparatus with two or more terminals that have the same terminal designation. (refer to “Connection point”).

Title block
This part of a form is often placed in the bottom right corner of the drawing frame. It contains administrative information about the drawing.

Categories and Types

APPENDIX A

The symbol directory grouping

Symbols for diagram

Group	Description
coils	Coils
conduct	Conductors
connect	Connectors
contacts	Contacts
contblk	Contact block
filters	Combination of inductors, condensator resistors
i_o_unit	Digital unit , I/O
inductor	Inductors
instrum	Measuring instruments
meascirc	Measuring circuits
motors	Motors
passcomp	Passive components
pilotsw	Pilot-switches
pow_circ	Power circuits
primtve	Primitive symbol

Group	Description
protect	Protection device
qualif	Qualifying symbol acc. to IEC617-2
semicond	Semi-conductors
sensors	Sensors, non electrical input
signal	Signal device
single	Single line
source	Current and voltage source
supply	Supply
switch	Switches, mechanical or static
transduc	Transducers
transf	Transformers
trmblk	Terminals, auxiliary circuits
trmbox	Terminals, boxes
trmmain	Terminals, main circuits
units	Function units, function groups
xref	Cross reference
applik	Application specific symbols
logic	Logic symbols

Symbols for layout

Symbol type	Library	Level	Note
Enclosure	CUBICLE	0	
Mounting plate	PLATE	2	
Device group	GROUPS	2	
Panel instrument	INSTR	6	Front-mounted apparatus
Rotating switch	ROTSW	6	
Push-button	PUSHBUTT	6	
Lamp/Indicator	LAMP	6	

Symbol type	Library	Level	Note
Front-mounted, other	MSC-PAN	6	
Terminal blocks	TERMINAL	6	Cubicle-mounted apparatus
Contactor	CONTACT	6	
Auxiliary relay	AUXRELAY	6	
Control system	PLC	6	
Breaker	BREAKER	6	
Fuse	FUSE	6	
Front-mounted, other	MSC-CUB	6	
Mounting-bars	BARS	4	These symbols are so called 1x1-block, with correct height and 1mm width.
Current- nd earth-rail	RAILS	4	
Cable-duct	DUCTS	4	
Cubicle-base	BASE	0	Accessories etc.
Hole punch	HOLES	6	
Label/Sign	LABEL	6	
Additional symbol	ADDITION	6	
Ruler	RULER	0	
Accessories, other	MSC-ADD	-	

Symbol categories

When inserting, the symbol category is retrieved from the CATEGORY1 field in the symbol database. If noting else is stated, the pre-selected object category is used.

Symbol category	Description	Pre-selected object category
2-WAY	2-way contact	RELAY
CAPACITOR	Capacitor	CAPACITOR

Symbol category	Description	Pre-selected object category
CHANGE_OVER	Change-over contact	RELAY
COIL	Coil	RELAY
CONNECT	Connector	CONN_UNIT
FILTER	Filter. A combination of passive components	FILTER
I/O_CHANNEL	I/O channel for control system	IO_CHANNEL
INDUCTOR	Inductor, coil, reactor	HIGH_VOLT_DEV
INSTRUMENT	Measure instrument	INSTRUMENT
MOTOR	Motor	MOTOR
NORMALLY_CLOSED	Normally closed contact	RELAY
NORMALLY_OPEN	Normally open contact	RELAY
PILOTSW	Measuring relay	PILOT_SWITCH
POW_CIRC	Switching device	CIRQUIT_BREAKER
PROTECT	Fuse	FUSE
RESISTOR	Resistor	RESISTOR
SEMICOND	Semi-conductor	HIGH_VOLT_DEV
SENSOR	Measuring device	TRANSMITTER
SIGNAL	Signal device	SIGNAL_DEV
SOURCE	Current- voltage device	POWER_SUPPL
SWITCH	Switching device for control mechanism	CONTROL_SWITCH
TRANSDUC	Transducer	TRANSDUCER
TRANSFORM	Transformer	TRANSFORMER

Symbol types

When inserting, the symbol type is retrieved from the CATEGORY2 field in the symbol database. The prefix indicates the type of drawing that the code is intended for. “EA” means layout and “ED” means diagram.

Symbol type	Description
EA_RAIL	Mounting rail. A symbol with an adjustable x-scale
ED_MAIN	Main symbol for an apparatus that has been drawn separated into smaller parts. Controls cross-references.
ED_SIG	Signal, general
ED_SIGSINK	Signal, source
ED_SIGSRC	Signal, sink
ED_TERMBLK	Terminal
ED_XBLK	Cross-reference symbol. For insertion of a reference to an apparatus that has been drawn on another sheet.

Attributes in symbols

TAG	Description
ID1-4	Designations
1A-9A	Item designations
*A	Shared item designation in e.g. a terminal
REM1-2	Remarks
CREF	Cross-reference in a main symbol
REF1-9	Cross-reference in a reference symbol

Extended Data

The REGAPP used for data in symbols is SOSED_SYMBDATA

TAG	Description
FUN.1	Function designation
LOC.1	Location designation
PRO.1	Product designation
SIG.1	Signal
TRM.1	Terminal 1
TRM.2	Terminal 2
TRM.n	Terminal n
DBCAT	Object category
OBJECTTYPE	Object type
SYMBOLTYPE	
DBOBJID	
REFHANDLE	

Configurations

APPENDIX B

Basic configuration

The following layers are generated when a new drawing is created:

Layer name	Status	Color	Line type
SOSED_CIRCUIT	On	7(white)	CONTINUOUS
SOSED_CIRCUITOFF	Off	1 (red)	CONTINUOUS
SOSED_DECORATIVE	On	4 (cyan)	CONTINUOUS
SOSED_DECORATIVEOFF	Off	1 (red)	CONTINUOUS
SOSED_DESIGN	Off	2 (yellow)	CONTINUOUS
SOSED_DESIGNERROR	Off	1 (red)	CONTINUOUS
SOSED_DESIGNREVIS	On	1 (red)	CONTINUOUS
SOSED_DOCFRAME	On	7 (white)	CONTINUOUS

The active layer is: SOSED_CIRCUIT

*el*Master Diagram

The following settings will be saved in the drawing:

Variable	Variable name	Valid values	Note
Height of terminal designation	TERMINAL_TEXTHEIGHT	NUMBER	
Standard for item designations	ITEMDES_TYPE	IEC750/ NEWIEC/ DOK80	
Prefix	ITEMDES_PREFIX	0/1	
Level separation	ITEMDES_SEP	./Ó./ÓÓ/=+-	
Signal handling	ITEM_REF	'=/+/-	
Attribute for sheet number	SHEET_TAG	TAG	
Attribute for document ID	DOCID_TAG	TAG	
Fixed value for sheet number	SHEET_VALUE	Optional string	
Fixed value for document ID	DOCID_VALUE	Optional string	

The following setting will be used but not saved:

Variable	Variable name	Valid values	Note
Path to EPS-database	EPSDBPATH	Path	If no value is given, the value of the AutoCAD's variable DWGPREFIX will be used. The value will only be used from a local CFG-file e.g. SETTINGS.CFG in the same directory as the drawing and not from the file DEFAULT.CFG

*el*Master Layout

Variable	Variable name	Valid values	Note
Insertion of hole pattern	MALLINS	0/1	

Format

Variable	Variable name	Valid values	Remark
Format type	FORMATTYPE		Reads the value from Format-DB at insertion of a form
Layer for form	FORMATLAYER		Created when form is inserted
Library for formats	FORMAT_LIBRARY	Last path in MLUSER-tree	Reads the value from Format-DB at insertion of a form
Group for format	FORMSET	Library under FORMAT in MLUSER-tree	Reads the value from Format-DB at insertion of a form
Format name	FORMATNAME	DWG-file	Reads the value from Format-DB at insertion of a form
Automatic insert of form	AUTOINSERT	0/1	Only used when a new drawing is created
Edit data in form when inserted	TB_MANEDIT	0/1	
Name of title block in form	HEADNAME	Blockname	
Automatic update of information in title block	AUTOUPDATE_FROM_DMI	0/1	
Automatic update of information document handling system	AUTOUPDATE_TO_DMI	0/1	
Revision head	REVBLOCK	Blockname	
Radius for revision ring	REVRADIUS	NUMBER	
Revisions list row	REVLISTROW	Blockname	Reads the value from Format-DB at insertion of a form

Variable	Variable name	Valid values	Remark
Number of rows	REVMAX	whole number	Reads the value from Format-DB at insertion of a form
Insertion point for revision list row	REVPOS	X,Y	Reads the value from Format-DB at insertion of a form
Distance between revision list rows	REVOFFSET	X,Y	Reads the value from Format-DB at insertion of a form
Size of reference grid	GRIDSIZE	X,Y	Reads the value from Format-DB at insertion of a form
Number of reference grids	GRIDREF	X,Y	Reads the value from Format-DB at insertion of a form
Offset for reference net	GRIDPOS	X,Y	Reads the value from Format-DB at insertion of a form
Sign for reference net	GRIDCHAR	X,Y	Reads the value from Format-DB at insertion of a form

Layer

Variable	Variable name	Valid values	Remark
Lock non-active layer groups	LGRAUTOLOCK	0/1	

Attribute handling

Variable	Variable name	Valid values	Remark
Handling of attributes at insertion of block	I_ATTRIB_EDITMODE	0-5	

Dimension setting

Variable	Variable	Valid values	Remark
Distance between object and dimension line	MDIM_MLDIMEXO	NUMBER	
Text parallel with line	MDIM_PARATEXT	0/1	

Item list

Variable	Variable name	Valid values	Remark
Block name	POSBLOCK	Blockname	
Radius	POSRADIUS	NUMBER	
Shape	POSSHAPE	0/1	
Type of connection	POSENDBLOCK	AUTO/ARROW/DOT/DASH	
Arrow block	ARROWHEAD	Blockname	
Title row block name	ITEMLISTHEAD	Blockname	
Row block name	ITEMLISTROW	Blockname	
Template	ITEMLISTTEMPLATE	Blockname	
Dist. title row	ITEMLISTDISTHEAD	NUMBER	
Dist. row	ITEMLISTDISTROW	NUMBER	
Columns to sum	ITEMLISTSUMCOLUMNS	NUMBER, NUMBER, NUMBER	
Sum columns	ITEMLISTSUM	0/1	

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